7 Features

This chapter describes the functions available through the Feature Icon Bar.
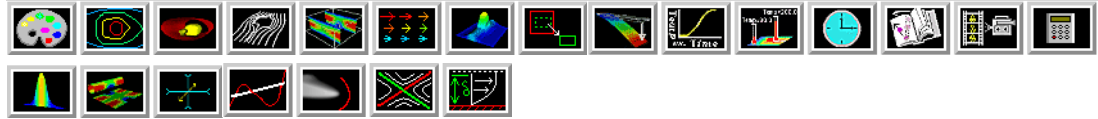

Figure 7-1
EnSight Feature Icon Bar

7.1   Color

Clicking once on the Color Icon opens the Color Editor in the Quick Interaction Area which allows you to assign color to the individual Part(s) which has(have) been selected in the Parts List. If no Parts are selected, modifications will affect the default Part color and all Parts subsequently loaded or created will be assigned the new default color.
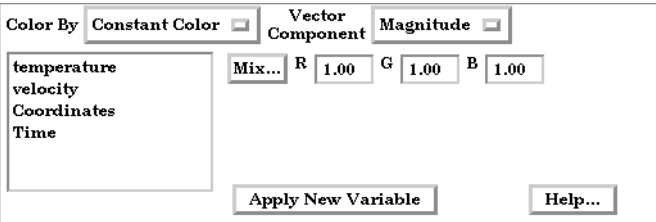

Figure 7-2
Color Icon

Figure 7-3
Quick Interaction Area - Color Editor - Constant Color

Color By   Opens a pull-down menu which allows you to choose whether to color the selected Part(s) by a Constant Color or by the Variable selected in the Variables List.

Constant Color   The selected Part(s) may be assigned a constant color in two ways. First, the color may be assigned by entering red, green, and blue numerical values (0.0 to 1.0) in the R,G,B fields of the Quick Interaction Area and then pressing the return key.

Mix...   Second, you can click on the Mix... button and the Color Selector dialog will open.
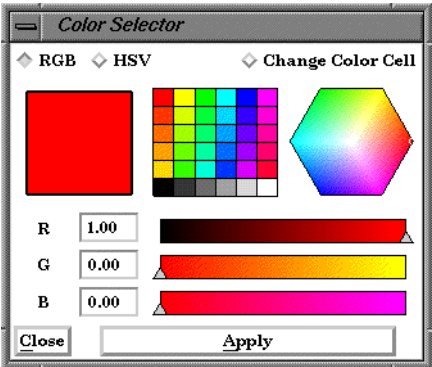
Figure 7-4
 Color Selector dialog

You can choose whether you wish to use the RGB color scheme or HSV. A color may be chosen in one of four ways. First, a color may be chosen from one of the color cells (small squares of constant color). Second, you can grab the small circle in the color assignment hexagon and interactively pick the desired color. Third, you can pick a color by entering numerical values (0.0 to 1.0) in the numerical (RGB or HSV) fields and then pressing return key. Finally, you can interactively choose a color by using the slider bars to the right of the numerical (RGB or HSV) fields which adjust the values therein (the color in the slider bars indicates the effect that modifying the color components will have on the final color).

It is possible to assign to the Color Cells area a color that you have specified in one of the other three ways by clicking the Change Color Cell and then clicking on the cell to which you wish to assign the currently defined color (as shown in the large rectangle to the left). This reassignment will be retained for use in subsequent EnSight sessions.

Regardless of which method you use to define a color, it will not be applied to the selected Part(s) unless and until you click the Apply button.
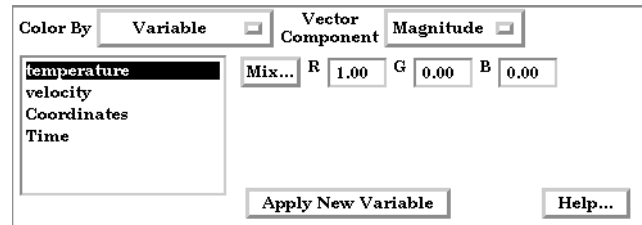

Figure 7-5
Quick Interaction Area - Color Editor - Variable

Variable	Alternatively, the Part(s) may be colored by a variable selected in the Variables List instead of by one constant color. The color palette for each Variable associates a color with each value of the variable and these colors are used to color the selected Part(s).
Vector Component	If you are coloring by a vector variable, this opens a pull-down menu which allows you to choose whether you wish to color using the magnitude or a component of the vector.
Magnitude	Color by the vector magnitude.
X	Color by the vector's X component.
Y	Color by the vector's Y component.
Z	Color by the vector's Z component.
Apply New Variable	Changes the color palette used to color the selected Part(s) to that of the variable currently chosen in the Variables List. If more than one variable is selected, then the color palette of the first selected variable will be used.

Feature Detail Editor (Variables)

Double clicking on the Color Icon will open the Feature Detail Editor (Variables) dialog.

(see Section 4.1, Variable Selection and Activation, Section 4.2, Variable Summary & Palette, and How To Edit Color Palettes)

7.2 Contour Create/Update

Contours are lines that trace out constant values of a variable across the surface(s) of selected Part(s), just like contour lines on a topographical map.
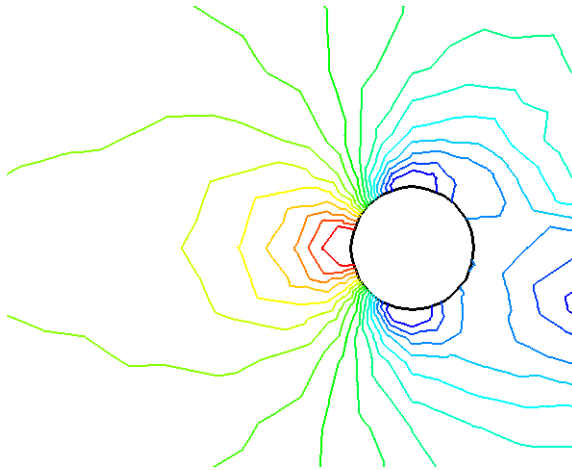

Figure 7-6
Pressure Contours in a Flow Field around a Circular Obstruction

The variable must be a node-based scalar, but can, of course, be a function of a vector variable (i.e., the magnitude or a component). A Contour Part can consist of one contour line, or a set of lines corresponding to the value-levels of the variable palette. A Contour Part has its own attributes independent of those Parts used to create it (the parent Part(s)).

Contours are drawn across the faces of parent Part elements (one-dimensional elements are ignored). At each node along the edges of any one element face, the contour's variable has a value. If the range of these values includes the contour's value-level, the contour line crosses the face. EnSight draws the contour by dividing the face into triangles each having the face's centroid as one vertex. For each triangle the contour crosses, it will cross only two sides. EnSight interpolates to find the point on each of those two sides where the variable value equals the contour value-level, then creates a bar element to connect the two points. Note that a contour line can bend while crossing an element face.

Because Contour Parts are created on the EnSight Client, the Representation attribute of the parent Part(s) greatly affects the result. Representations that reduce Part elements to one-dimensional representations (Border applied to two-dimensional Parts and Feature Angle), or do not download the Part (Not Loaded), will eliminate those Part elements from the Contour creation process. On the other hand, Full representation of three-dimensional elements will create contour lines across hidden surfaces. Usually, you will want the Representation selection to be 3D Border, 2D Full.

Contour Parts are created on the Client, and so cannot be queried or used in creating new variables. However, Contours can be used as parent Parts for Profiles and Vector Arrows.

If you change the value-levels in the Feature Detail Editor (Variables) Summary and Palette section, the Contour automatically regenerates using the new value-levels.

Use care when simultaneously displaying contours based on different function palettes so that you do not become confused as to which contours are which. Coloring them differently and adding an on-screen legend can help.

Clicking once on the Contour Create/Update Icon opens the Contour Editor in the Quick Interaction Area which is used to both create and update (make changes to) contour Parts.
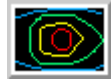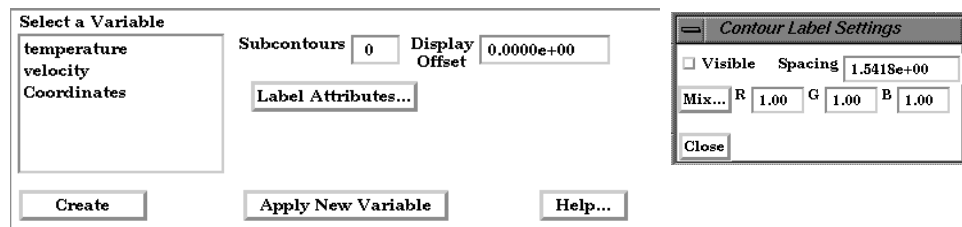

Figure 7-7
Contour Create/Update Icon

Figure 7-8
Quick Interaction Area - Contour Editor

Subcontours	This field allows you to specify the number of sub-contours you wish to be drawn at evenly spaced value-levels between the value-levels defined in the Variable Feature Detail Editor Summary and Palette section. Leaving this field 0 will produce exactly the number of contour lines for which value levels are specified in the Feature Detail Editor (Variables) Summary and Palette.
Display Offset	This field specifies the distance away from a surface to display the contours (so that potential Z-buffer conflicts can be overcome). A positive distance value moves the contour away from the surface in the direction of the surface normal.
Labels	
Visible Toggle	Toggles on/off the visibility of number labels for contour lines.
Spacing	Determines the spacing between labels.
Mix...	Opens the Color Selector dialog for the assignment of a color to labels.
R,G,B	Allows the specification of red, green, and blue values for the assignment of a color to labels.
Create	Creates a Contour Part using the selected Part(s) in the Parts List and the color palette associated with the Variable currently selected in the Main Variables List.
Apply New Variable	Will change the Contour Part to show contours based on the color palette associated with the Variable currently selected in the Variables List.
Vector Component	If the selected Variable is a vector, this option allows you to choose between the vector magnitude or the X, Y, or Z component.
Feature Detail Editor (Contours)	Double clicking on the Contour Create/Update Icon opens the Feature Detail Editor (Contours), the Creation Attributes Section of which provides access to the same functions available in the Quick Interaction Area. For a detailed discussion of the remaining Feature Detail Editor turn-down sections (which are the same for all Part types):

(see Section 3.3, Part Editing and How to Create Contours)

Troubleshooting Contours

Problem	Probable Causes	Solutions
No contours created.	Variable values on element faces are outside range of palette function value-levels.	Adjust palette function using the Feature Detail Editor (Variables) Summary and Palette section.
	Parent Parts do not contain any 2D elements.	Re-specify Parent Part list.
	Parent Parts do not contain the specified Variable.	Recreate the Variable for the selected Parent Part(s).
Too many contours.	Palette has too many function levels.	Change the number of levels for the palette using the Feature Detail Editor (Variables) Summary and Palette.
	Specified too many sub-contours.	Lower the sub-contour attribute.
Too few contours.	The palette levels do not adequately cover the function value range for the Parent Parts.	Modify the palette using the Feature Detail Editor (Variables) Summary and Palette.
	Sub-contour attribute set to 0.	Modify the Sub-contour attribute.
Contour Part created but (empty)	Parent Part is in Feature Angle representation.	Change Parent Part to 3D border, 2D full representation.
Contours are fine at first, but later go away.	Parent Parts representation changed to Feature Angle, or Not Loaded.	The contours are created from the Part representation on the EnSight client. Modifying the representation affects the Contour Parts.

7.3 Isosurface Create/Update

Isosurfaces are surfaces that follow a constant value of a variable through three-dimensional elements. Hence, isosurfaces are to three-dimensional elements what contour lines are to two-dimensional elements.
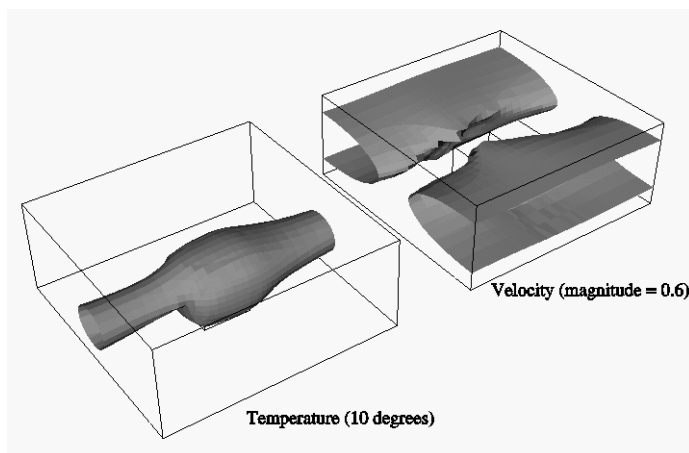

Figure 7-9

An isosurface may be based on a vector variable (magnitude or components), or a scalar variable.

At each node of a three-dimensional element, the isosurface's variable has a value. If the range of these values includes the isosurface's isovalue, the isosurface cuts through the element. EnSight draws the isosurface through that element by first determining which edges the isosurface crosses, and then interpolating to find the point on each of those edges corresponding to the isovalue. EnSight connects these points with triangle elements passing through the parent Part elements. If the Parent Part(s) contain two-dimensional elements, a line is created across the elements - just like a contour.

All the triangle elements created inside all the three-dimensional elements of all the parent Part(s) together with all the lines created across the two-dimensional elements of all the Parent Part(s) constitute the isosurface. One-dimensional elements of the parent Part(s) are ignored. Because isosurfaces are generated by the server, the Representation of the parent Part(s) is not important.

You can interactively manipulate the value of an isosurface with a slider allowing you to scan through the min/max range of a variable. This scanning can also be done automatically. The isosurface will change shape as the value is changed.

If you are using animation, you can specify an Animation Delta value by which the isovalue is incremented for each animation frame or page. The isosurface is automatically updated to appear as if it had been newly created at the new location and time.

Clicking once on the Isosurface Create/Update Icon opens the Isosurface Editor in the Quick Interaction Area which is used to both create and update (make changes to) isosurface Parts.
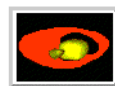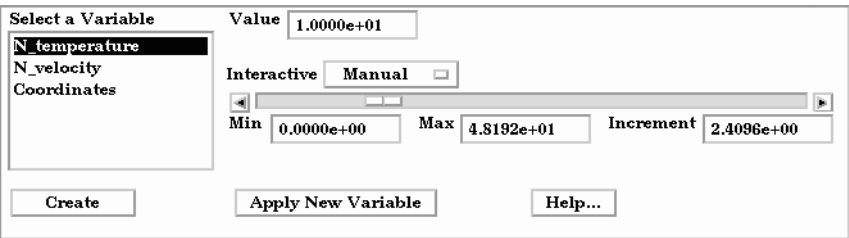

Figure 7-10
Isosurface Create/Update Icon

Figure 7-11
Quick Interaction Area - Isosurface Editor

Value	Specification of numerical isovalue of the isosurface. To avoid an empty Part, this value must be in the range of the Variable within the Parent Parts. You can find this range using the Variables dialog or by showing the Legend for the Variable. For vector-variable-based isosurfaces, the vector magnitude is used.
Interactive	Opens pull-down menu for selection of type of interactive manipulation of the isosurface value. Options are:
Off	Interactive isosurfaces are turned off.
Manual	Value of the isosurface(s) selected are manipulated via the slider bar and the isosurface is interactively updated in the Graphics Window to the new value.
Auto	Value of the isosurface is incremented by the Auto Delta value from the minimum range value to the maximum value when the cursor is moved into the Main View. When reaching the maximum it starts again from the minimum.
Auto Cycle	Value of the isosurface is incremented by the Auto Increment value from the minimum range value to the maximum value. When reaching the maximum it decrements back to the minimum.
Auto Delta	Specification of the increment for the Auto and Auto Cycle options to use when modifying the value between the minimum and maximum values.
Min	Specification of the minimum isosurface value for the range used with the "Manual" slider bar and the "Auto" and "Auto Cycle" options.
Max	Specification of the maximum isosurface value for the range used with the "Manual" slider and the "Auto" and "Auto Cycle" options.
Increment	Specification of the increment/decrement the slider will move within the min and max, each time the stepper buttons are clicked.
Create	Creates an isosurface Part at the value specified for the variable selected in the Variables List and from the Part(s) selected in the Parts List.
Apply New Variable	Will recreate the isosurface Part at the value specified for the variable currently selected in the Variables List.

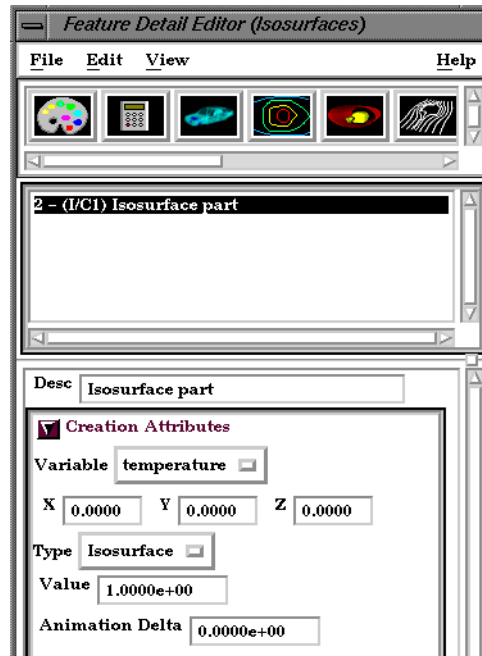| Feature Detail Editor (Isosurfaces)	Double clicking on the Isosurface Create/Update Icon opens the Feature Detail Editor (Isosurfaces), the Creation Attributes Section of which provides access to additional features for isosurface creation and modification:

Figure 7-12
Feature Detail Editor (Isosurfaces) Creation Attributes Area

Variable	Opens a pop-up menu for the selection of an active Variable to use to calculate the isosurface.
X Y Z	These fields specify the vector- component coefficients. When the three fields are set to 0.0000, the vector magnitude is used for the isosurface calculation. Otherwise, the sum of: $(Vector_x * X) + (Vector_y * Y) + (Vector_z * Z)$ is used as the isosurface value.
Type	
Isosurface	Specification that an Isosurface type part created from the specified Variable and selected parts will have the isovalue of Value for all its elements.
Value	Specification of the numerical isovalue of the Isosurface Part(s) selected in the Feature Detail Editor's Parts List (or if none is selected, of the isosurface you are about to Create).

Figure 7-13
Feature Detail Editor (Isovolume) Creation Attributes Area

Isovolume	Specification that an Isovolume type part created from the specified Variable and selected parts will consist of elements with isovalues constrained to either below a Min, above a Max, or within the specified interval of Min and Max.
Constraint	Specification restricting the element isovalues of the Isovolume Part to an interval. The Constraint options are:
	Low all elements of Isovolume Part have isovalues below the specified Min value.
	Band all elements of Isovolume Part have isovalues within the specified Min and Max interval values.
	High all elements of Isovolume Part have isovalues above the specified Max value.

Min	Specification of the minimum isovalue limit for the Isovolume Part.
Max	Specification of the maximum isovalue limit for the Isovolume Part.

Animation Delta This field specifies the incremental change in isovalue for each frame or page of animation It can be negative.

(see Section 7.14, Flipbook Animation and Section 7.15, Keyframe Animation)

Create (At the bottom of the Feature Detail Editor) Creates an Isosurface Part at the value specified for the variable selected in the Variable pop-up menu of the Creation Attributes section and from the Part(s) selected in the Main Parts List.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

For a detailed discussion of the remaining Feature Detail Editor turn-down sections (which are the same for all Parts):

(see Section 3.3, Part Editing and How To Create Isosurfaces

7.4   Particle Trace Create/Update

A *Particle trace* visualizes a vector field by displaying the path that a massless Particle would follow if placed in that field. At each point on the Particle trace, the direction of the trace is parallel to the vector field at that point and time.

A *streamline* is a Particle trace in a steady-state vector field, while a *pathline* is a Particle trace in a time-varying vector field. Particle traces can be lines or "ribbons" (that additionally visualize the rotation of the vector field around the path of the trace).

EnSight is capable of computing a pathline through a model with changing coordinates and/or changing connectivity. The variable values are assumed to behave linearly between the known timesteps.

Particle Trace Parts have their own attributes, so you can, for example, trace a flow field using the velocity variable, and then color the resulting trace using the temperature variable.
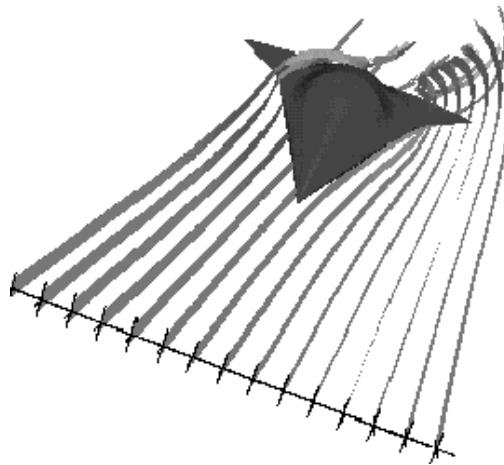

Figure 7-14
Particle Trace Illustration

Emitters

A Particle Trace Part consists of one or more Particle traces originating from points on one or more *emitters*. Each emitter is capable of emitting a Particle starting at a specified time and continuing to emit Particles at given intervals. When pathlines are generated with emitters emitting at multiple time intervals and these traces are then animated, *streaklines* are displayed.

Emitters consist of single points, points along a line, points forming a grid in a plane, or points corresponding to the nodes of a Part. You can define emitters using the Cursor tool, the Line tool, the Plane tool, or a Part.

Emitters can be created using the cursor, line, and plane tools, using existing Part nodes, or can be created in a surface restricted mode where the mouse can be used to project points, rakes or nets directly onto the displayed surfaces of the model.

Pathlines, of course, must be drawn forward in time, but streamlines can be drawn forward in time, backward in time, or both. Each Particle trace terminates when either (1) the Particle trace moves outside the space in which the vector field is defined, (2) a user-specified time limit is reached, (3) the massless Particle becomes stationary in a place where the vector field is zero, or (4) the last transient-data time step is reached. (4 applies only to pathlines)

A Particle trace can pass through any point inside an element of the parent Part(s). The vector field at any point is calculated from the shape function of the containing element. Emitter points located outside the elements are ignored when creating Particle traces.

Surface-Restricted Traces

A surface-restricted Particle trace is constrained to the surface of the selected Part(s) by using only the tangential component of the velocity. The velocity values for this type of trace can be the velocity at the surface (if nonzero) or at some user specified offset into the velocity field.

Interactive Traces

A Particle trace can be updated interactively by entering interactive mode and moving the tool used to create the emitter. When a trace is selected and interactive emitter is turned on, the tool will appear at the location of the emitter. The user then manipulates the tool interactively in the Graphics Window or using the transformations dialog. (This option is not available for surface-restricted Particle traces or traces emitted from a Part).

Integration Method

EnSight creates Particle traces by integrating the vector field variable over time using a Fourth Order Runge-Kutta method and utilizing a time varying integration step. The integration step is lengthened or shortened depending on the flow field, but you can control the minimum number of integration steps performed in any element as well as other time step controls.

Normally, EnSight will perform the integration using all of the components of the vector. However, it is possible to restrict the integration to a plane by specifying which components of the vector to use. Typical uses of this feature would be to restrict the Particle traces to a clip plane. Surface-restricted Particle traces provide even greater flexibility in restricting a trace to planes or other surfaces.

Line-type Particle traces consist of bar elements. Ribbons consist of 4-noded quad elements and originate with their end-edge parallel to the Z-axis of the global frame. Then, at each integration step, the leading edge is rotated around the current direction of the path by the same amount the vector field has rotated around the path since the previous time step. Ribbons are not available for surface-restricted Particle traces.

Particle Trace Parts are created on the server, so the Representation-type of the parent Parts has no effect. The algorithm that creates Particle traces initially sets up a cross-referencing map of adjoining elements. Hence, the first Particle trace takes longer to generate than subsequent traces.

If you calculate pathlines, consider calculating as many as possible at a time, since the process can be very time consuming (most of the time is taken in reading time step information). However, the data for the Trace Part is sent to and stored on the Client, Thus, you cannot label or make queries about Particle Trace Parts. Instead, label or make queries about the Particle Trace's parent Part(s). Line-type Particle Traces can be parent Parts for Profiles. You can animate the motion of the massless Particles along their Particle traces.

Transient Data

By default the emission point is always set to emit the Particles at the current time step. This can be a problem if you have a transient dataset with the current time set at the last time step available. If you compute pathlines from this location, the default emission time will be at the current time (last time step), thus no pathlines will be generated. To solve this problem you will need to either change the current time, or change the Start Time of the emitter.

The process of creating a Particle trace is always to specify an emission point (location and time), specify the Part(s) to trace the Particle through and specify which vector variable to integrate. There are quick ways of doing this process which assume that the correct defaults are set, or there are more deliberate ways which give you more control. Particle trace Parts carry only one set of attributes for all of the traces in the Part, thus it is not possible, for example, to trace some of the emission points forward in time and others backward in time.

Particle trace Parts are different from all other created Parts in that when the parent Parts change (such as at a time step change), the Particle trace Part does not change. This is due to the fact that the Particle trace has been created at a specified time (the emission time), making the Part independent of time (after the trace has been created).

Regular Particle traces can only be computed through a set of parent Parts consisting of model Parts. Surface-restricted Particle traces can be created on model Parts, clip Parts, elevated surface Parts, and developed surface Parts.

If your dataset contains 3D elements, the Particles for regular traces will be traced through 3D element fields only. Surface-restricted traces would have to be used to trace along 2D elements of such a data set.

Massed-Particle Traces A particle trace can be created or updated from a massless-particle trace to a massed-particle trace, or visa-versa. Massed-particle traces are specified via their appended section in the Feature Detailed Editor (Traces) dialog. Massed-particle traces switch to massless-particle traces during interactive mode.

Definitions

Motion of a particle as a function of its velocity is defined as

$$\texttt{d/dt}(\texttt{X}_\texttt{p}) \ = \ \texttt{V}_\texttt{p}$$

with initial conditions $\texttt{V}_\texttt{p}(\texttt{t}_0)=\texttt{V}_{\texttt{p},0}$ and initial particle position $\texttt{X}_\texttt{p}(\texttt{t}_0)=\texttt{X}_{\texttt{p},0}$ (capital letters denote vectors unless otherwise indicated).

For massless particles, the particle velocity is always identical to the local fluid velocity, $\texttt{V}_\texttt{p}=\texttt{V}_\texttt{f}$. For massed particles, additional forces acting on them result in a different velocity for the $\texttt{V}_\texttt{p}$ than for the fluid, $\texttt{V}_\texttt{p}$ not equal to $\texttt{V}_\texttt{f}$. This particle velocity is determined from a momentum balance for the particle by

$$\texttt{m}_\texttt{p} \, \texttt{A}_\texttt{p} = \texttt{F}_\texttt{p},$$

or

$$\texttt{m}_\texttt{p} \ \texttt{d/dt}(\texttt{V}_\texttt{p}) \ = \ \texttt{F}_\texttt{g} \ + \ \texttt{F}_\texttt{p} \ + \ \texttt{F}_\texttt{d} \ + \ \texttt{F}_\texttt{e},$$

where

$\texttt{A}_\texttt{p}$ = particle acceleration vector,

$\texttt{F}_\texttt{p}$ = total (particle) force vector,

$\texttt{F}_\texttt{g}$ = gravitational (body) force vector = $\texttt{m}_\texttt{p} \ \texttt{b} \ \texttt{G},$

$\texttt{F}_\texttt{p}$ = pressure (surface) force vector = $- \ \texttt{v}_\texttt{p} \ \nabla \texttt{P}_\texttt{f},$

$\texttt{F}_\texttt{d}$ = drag (surface) force(s) vector = $\frac{1}{2} \ \rho_\texttt{f} \ \texttt{a}_\texttt{p} \ \texttt{c}_\texttt{d} \ |\texttt{V}_\texttt{r}| \ \texttt{V}_\texttt{r},$

$\texttt{F}_\texttt{e}$ = additional forces vector, here = $0,$

given the following definitions (Note: the <u>underlined</u> definitions are user specified)

m_p = particle mass = $\rho_p \, v_p$,

ρ_p = <u>particle density</u>,

v_p = particle volume = $d_p^3 \, \pi/6$,

d_p = <u>particle diameter</u>,

b = particle buoyancy ratio = $(\rho_p - \rho_f)/\rho_p$,

ρ_f = <u>fluid density (scalar or constant)</u>,

G = <u>gravitational acceleration vector</u>,

∇P_f = <u>fluid pressure gradient vector</u>, (computed from P_f = fluid pressure scalar variable)

a_p = particle reference area = $d_p^2 \, \pi/4$,

V_r = reference velocity vector = $V_f - V_p$,

c_d = drag coefficient, typically given as a function of the local relative Re, i.e. $c_d = \underline{c_d(Re)}$,

Re = Reynolds number = $\rho_f \, |V_r| \, d_p \, / \, \mu_f$,

μ_f = <u>fluid dynamic viscosity</u> (scalar or constant).

Thus, the total mass balance equation for massed particles may be defined by:

$m_p \; d/dt(V_p) \; = \; (m_p \; b \; G) \; - \; (v_p \; \nabla P_f) \; + \; (\tfrac{1}{2} \; \rho_f \; a_p \; c_d \; |V_r| \; V_r).$

Drag Coefficient

Currently, the following Drag Coefficient (C_d) table is provided as the default.

Re << 1 $C_d = 24/Re$

1 < Re << 500 $C_d = 24/Re^{0.646}$

500 < Re << 3e5 $C_d = 0.43$

3e5 < Re << 2e6 $C_d = 3.66E\text{-}4 \; Re^{.4275}$

2e6 < Re $C_d = 0.18$

This table is also coded as an example for your reference and access via the User-Defined Math Function `DragCoefTable1(Re)` which is found in

$CEI_HOME/ensight74/user_defined_src/math_functions/drag_coef_table1/libudmf-drag_coef_table1.c

In addition, two other drag coefficient functions are provided for your selection via the User-Defined Math Function facility.

`DragCoefPoly(Re)` $= (a + b \; Re + c \; Re^2 + d/Re)$

Where: {a,b,c,d} are polynomial coefficients with default values of {1.,0.,0.,0}, respectively.

`DragCoefPower(Re)` $= (1 + .15 \; Re^{0.687}) \; 24 \; / \; Re$

Both of these functions are located respectively in

$CEI_HOME/ensight74/user_defined_src/math_functions/drag_coef_poly/libudmf-drag_coef_poly.cf

$CEI_HOME/ensight74/user_defined_src/math_functions/drag_coef_power/libudmf-drag_coef_power.c

You may also code your own. (See UDMF in EnSight User Manual.)

Particle-Mass Scalar on Boundaries

Information to compute a particle-mass scalar on boundaries ($m_P = \sum m_{Pi}$) is provided each time massed-particle traces are created. This scalar is found and computed via the New Computed Variables (NCV) functionality.

Massed Particle Scalar(massed-particle traced part(s))

This scalar creates a massed-particle per element scalar variable for each of the parent parts of the massed-particle traces. This per element variable is the mass of the particle times the sum of the number of times each element is exited by a mass-particle trace.

References

The following references have contributed in part toward the development of the massed-particle algorithm.

Donley, H. Edward
  "The Drag Force on a Sphere",
  http:\\www.ma.iup.edu/projects/CalcDEMma/drag/drag.html

Lund, Christoph
  "Vorgaben für die Berechnung und Visualisierung der Bahnlinien massebehafteter Partikel im Postprozessor EnSight", Volkswagen AG, 27.07.2001.   English translation by Kent Misegades.

Fluid Dynamics International, Inc.
  FIDAP 7.0 Theory Manual", April 1993, pp12-3+

Clicking once on the Particle Trace Create/Update Icon opens the Particle Trace Editor in the Quick Interaction Area which is used to both create and update (make changes to) Particle trace Parts.
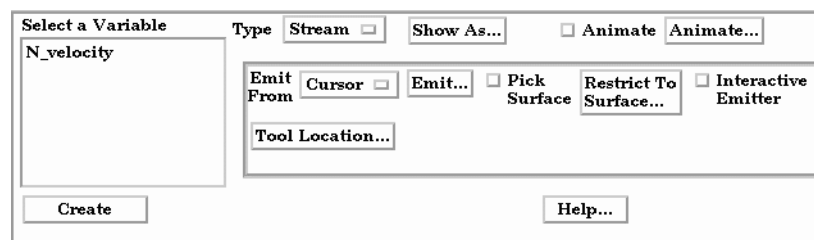

Figure 7-15
Particle Trace Create/Update Icon

Figure 7-16
Quick Interaction Area - Particle Trace Editor

Type   Opens a pull-down menu for specification of whether Particle trace calculation uses steady-state data (streamlines) or transient data (pathlines).

Stream	Traces a massless Particle in a steady-sate vector field (for steady-state data or the current time-step of transient data).
Path	Traces a massless Particle through a time-varying vector field *and so is only available with transient results data*. On certain systems, this selection can consume significant quantities of CPU time to calculate the resulting Particle trace.
Show As	Opens a dialog for specification of trace representation.
Line	Depicts the trace as a line.
Ribbon	Depicts trace as if it were a ribbon. The ribbon width is a specified fixed value, while the twisting is determined by the rotation of the flow about the path of the trace at any particular point on the trace.
Square Tubes	Depicts trace as if it were a square tube. The tube width is a specified fixed value, while the twisting is determined by the rotation of the flow about the path of the trace at any particular point on the trace.
Animate Toggle	Toggles on/off the animation of the motion of the Particles along the traces. In addition to creating Particle traces based on vector variables, EnSight can also animate the motion of the Particles along the Particle traces. To distinguish them from discrete Particles, we call Particles moving along Particle traces "tracers."

At any instant, each tracer consists of a portion of a Particle trace displayed with attributes you specify separately from the attributes of the Particle trace. EnSight animates each tracer by updating which portion of the Particle trace is currently displayed. You specify the length of each tracer as a time value, so the tracer's length varies dynamically as it moves down the Particle trace (faster moving tracers are longer). This option can add tremendously to the understanding of the flow field since relative speed can be determined.

EnSight provides control over how the tracer looks and acts. You can animate one, some, or all of the Particle traces you have created, but they are all animated in the one way you specify. To help you get started, at the click of a button EnSight will suggest time-specification values based on the Particle traces you have selected to animate. You can specify the line width of the tracer, and choose to color it with a constant color or the same calculated color used to color the Particle trace. You can also display a spherical "head" on the leading-end of the tracer, and dynamically size the head according to any active variable.

You control the speed of the motion and have the option to display multiple tracers on the same Particle trace separated by a time interval. Hence, you can choose to view rapid-fire pulses, slow moving "noodles," or something in between. For steady-state Particle traces (streamlines), "time" is the integration time with the emitters located at time zero. For transient Particle traces (pathlines), you have the option to synchronize the animation time to the solution time. The choice of whether a Particle trace is a streamline or a pathline is made when you create the Particle trace.

You do not have to animate the entire Particle trace. You can specify where you want the animation to start with a time value corresponding to a distance down the Particle trace from the emitter, and where you want the animation to stop with a time value corresponding to a distance farther down the Particle trace.

Tracers on all animated Particle traces are synchronized. If you combine Particle trace animation with flipbook animation or keyframe animation, the animation time values are automatically synchronized if you toggle-on Sync To Transient in the Trace Animation Settings dialog.

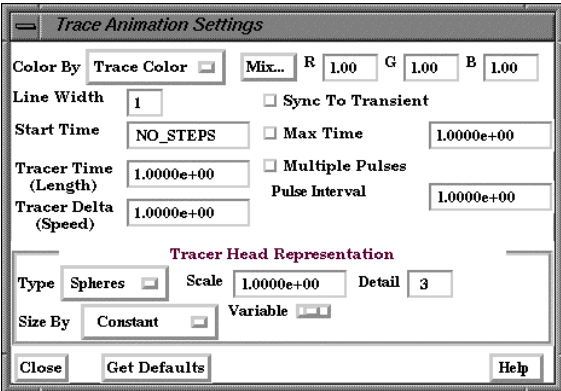|	
Animate...	Opens the Trace Animation Settings dialog

Figure 7-17
Trace Animation Settings dialog

Color By	Opens a pull-down menu for selection of method by which to color the tracers.
Constant	Displays tracers in the constant color specified in this dialog.
Mix…	Opens the Color Selector dialog (See Figure 7-4 Color Selector dialog).
R,G,B	Fields allow specification of constant color.
Trace Color	Displays tracers in the same color as the Particle Trace Part from which they originate.
Line Width	Specification of displayed width (in pixels) of tracers. Note: Line Width specification may not be available on some workstation platforms.
Start Time	Specification of how far down each Particle trace to begin displaying tracers. A Particle trace is made up of line segments. Each segment that makes up a Particle trace has an associated time value. The start time indicates where on the Particle trace the tracers will begin animation.
Tracer Time (Length)	Specification of length of tracers which varies as the tracer speed varies along the Particle trace. The Particle Time Length parameter scales the length of all tracers at all times.
Tracer Delta (Speed)	Specification of how fast tracers move. Longer times result in faster moving tracers. This parameter is not applicable when using Sync To Transient and displaying transient data through flipbook or keyframe animation.
Sync to Transient Toggle	Toggles on/off synchronization of tracer position to solution time of transient data. When toggled-on and transient data is in use, each tracer is displayed with its leading-end at the correct location along the Particle trace for the current solution time. Traces only move forward in time so cycling through transient data is not applicable here.
Max Time Toggle	Toggles on/off maximum lifetime for all tracers. If toggled-off, tracers continue to end of Particle trace. If toggled-on, each tracer stops after moving down the Particle trace for a distance corresponding to the specified Max Time (or until one of the other conditions that stop a tracer occurs).
Max Time	Field specifies lifetime of all tracers when Set Max Time is toggled-on.
Multiple Pulses Toggle	Toggles on/off multiple emission of tracers. When toggled-off, a single tracer for each Particle trace appears at the specified Start Time. When toggled-on, additional tracers appear after each specified Pulse Interval. Not applicable to pathlines.
Pulse Interval	Field specifies time delay between tracers. Not applicable when Multiple Pulses is toggled-off.

Tracer Head Representation

Type	Opens a pull-down menu for selection of type of head for each tracer.
None	Specifies that no head will appear.

Spheres	Specifies that a sphere will appear on the leading end of the tracer.
Scale	Specification of scaling factor for head size. Values between 0 and 1 reduce the size, factors greater than one enlarge the size. Not applicable when Head Type is None.
Detail	Specification of detail-level of head in range from 2 to 10, with 10 being the most detailed (e.g., rounder spheres because more polygons are used to create spheres). Higher values take longer to draw, slowing performance. Not applicable when Head Type is None.
Size By	Opens a pull-down menu for the selection of variable-type to use to size each tracer's head. If you select a variable, the head size is determined by multiplying the Scale factor times the variable value, which will vary depending on the location of the tracer. Not applicable when Head Type is None.
Constant	Sizes head using just the Scale factor value.
Scalar	Sizes head using a scalar variable.
Vector Mag	Sizes head using magnitude a vector variable.
Vector X	Sizes head using X-component of a vector variable.
Vector Y	Sizes head using Y-component of a vector variable.
Vector Z	Sizes head using Z-component of a vector variable.
Variable	Selection of variable to use to size the tracer heads. Not applicable when Type is None or Size By is Constant.
Get Defaults	Click to set time-value specifications in this dialog to values suggested by EnSight based upon the characteristics of the selected Particle traces.

See Also: How To Animate Particle Traces

Troubleshooting Animated Particle Traces

Problem	Probable Causes	Solutions
No motion. Can't see any tracers.	No Particle traces selected to animate.	Select the traces you wish to animate in the list at the top of the Animated Trace Setup dialog.
	Tracers colored same as Particle traces and have same line width.	Change Color By or Line Width.
	Animate Traces not toggled-on.	Toggle Animate on in the Quick Interaction Area.
	Start Time > maximum Particle trace time for all traces selected.	Change settings in the Trace Animation Settings dialog.
	Delta Time (Speed) set too high.	Change settings in the Trace Animation Settings dialog.
	Particle Time (Length) set too small.	Change settings in the Trace Animation Settings dialog.
Motion too fast.	Delta Time (Speed) set too high.	Change settings in the Trace Animation Settings dialog.
Can't get multiple pulses at same time.	Pulse interval too high.	Decrease to have pulses start closer together.
Have one big tracer, no pulses.	Pulse interval too small, pulses start right after each other with no separation.	Increase the interval.

Quick Interaction Area Particle Trace Editor, continued,

|	
Emit From	Opens a pull-down menu for the specification of the emitter type.
Cursor	Creates Particle trace beginning from the position of the Cursor tool.
Line	Creates Particle traces beginning from the position of the Line tool.
# Points	This field specifies the number of evenly spaced traces you want to emit from the Line tool.
Plane	Create Particle traces beginning from the position of the Plane tool.
# Points	These fields specify the number of traces you want to emit from the Plane tool in the X and Y axes of the tool.
Part	Creates particle traces beginning from each node of the Part specified by the Part ID Number field.
Part ID Number	This field specifies the Part you wish to use as an emitter for the creation of a particle trace. The Part ID Number for a Part is found in the Main Parts List. (see Section 3.1, Part Overview)
Emit...	Opens the Emission Detail Attributes dialog.
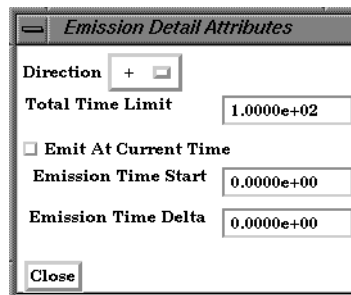
Figure 7-18
Emission Detail Attributes dialog

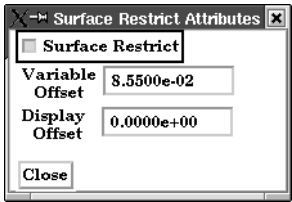|	
Direction	Trace the Particle in positive time, meaning to trace with the vector field, or trace the Particle in negative time, meaning to trace the Particle upstream. Option only applies to streamlines. Pathlines must be traced in + time.
(+)	Positive time option traces Particle(s) forward in time. (This is the only option for pathlines.)
(–)	Negative time option traces Particle(s) backward in time.
(+/–)	Positive/Negative time option traces Particle(s) both forward and backward in time.
Total Time Limit	This field specifies the maximum length of time the Particle trace may continue (it may terminate sooner for other reasons). For vector fields with recirculation zones, this can be important to keep from integrating a trace indefinitely.
Emission Time Start	This field specifies the simulation time at which to begin Particle emission. Enter value between beginning and ending time available.
Time Delta	This field specifies the time interval between emissions of Particles from the emitters. If "0", only one set of emissions will occur at start time
Pick Surface Toggle	Toggles on/off the feature which allows you to place the trace emitter at a point on a surface directly below the mouse pointer by clicking the left mouse button.
Restricted To	Opens the Surface Restricted Attributes dialog.

Surface . . .

Figure 7-19
Surface Restricted Attributes dialog

Surface-Restrict Toggle	Toggles on/off surface restricted feature for streamlines. The streamline will be constrained to stay on the surface of the selected Part(s) by using only the tangential component of velocity. Be sure to use the Pick Surface feature in locating the emitter for a surface restricted particle trace to ensure that the emitter is located on the surface of a Part.
Variable Offset	This field specifies the distance into the flow field at which velocity (and other variables) are to be sampled for the surface restricted trace(s). If velocity values are present at the surface, this offset can be set to zero.
Display Offset	This field specifies the distance away from the surface to display the surface restricted trace(s). This is provided so z buffer conflicts between surfaces and trace lines can be overcome. A positive distance moves the trace away from the surface in the direction of the surface normal, while a negative distance moves the trace in the opposite direction.
Interactive Emitter	Toggles on/off interactive Particle tracing. Manipulation of the Cursor, Line or Plane tool will cause the Particle trace to be recreated at the new location and updated in the Graphics Window. When manipulation of the tool stops, the Particle trace and any Parts that are dependent on it will be updated. (Only available for non-surface-restricted streamlines).
Tool Location...	Opens Transformations Editor dialog which allows you to precisely position the Cursor, Line or Plane tool.
Create	Creates a Particle trace Part using the selected Part(s) in the Main Parts List and the vector Variable selected in the Main Variables List.

Feature Detail Editor (Traces)   Double clicking on the Particle Trace Create/Update Icon opens the Feature Detail Editor for Particle Traces, the Creation Attributes Section of which provides access to additional functions for trace creation and modification:
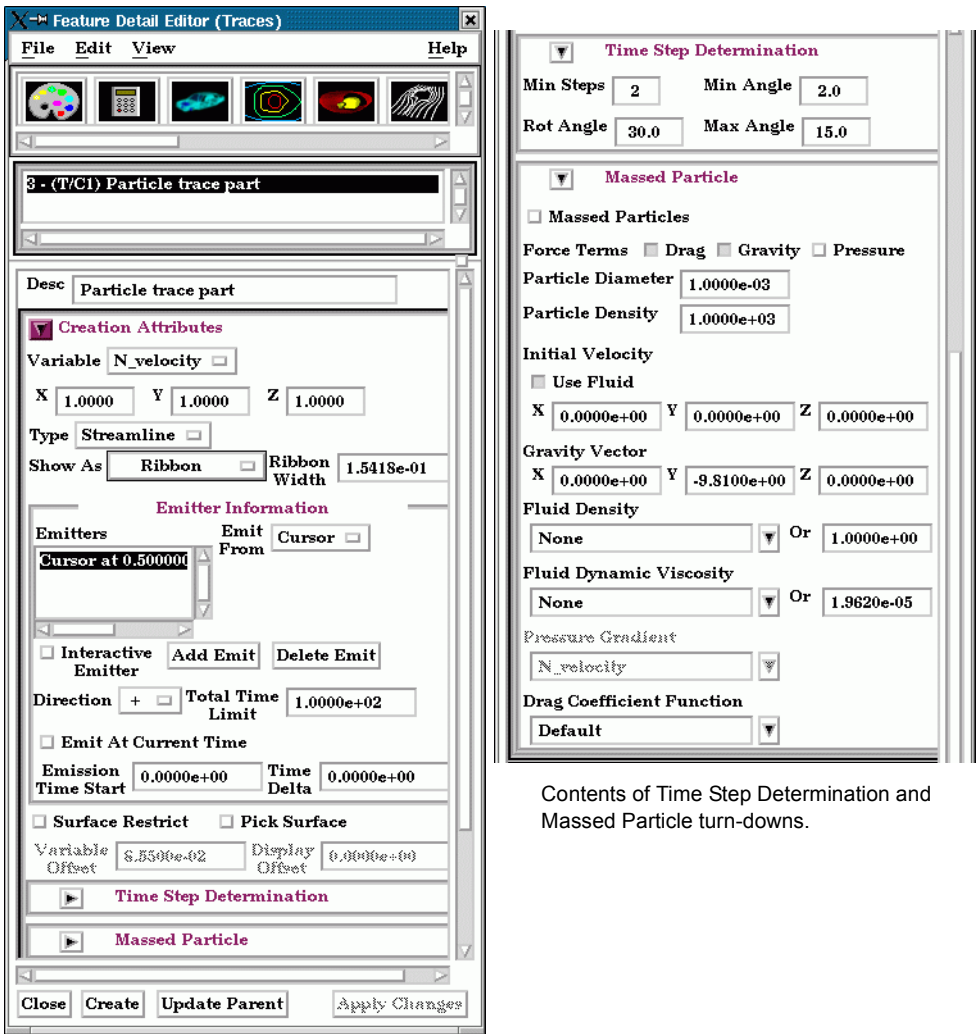
Contents of Time Step Determination and Massed Particle turn-downs.

Figure 7-20
Feature Detail Editor (Traces)

Variable	Opens a pop-up menu for the selection of an active variable to use to calculate the trace.
X Y Z	These fields specify the fraction of each vector component to be used in the calculation. Specify 1 to use the full value of the vector component. Specify 0 to ignore the corresponding vector component (and thus confine the motion of the Particle to a plane perpendicular to that axis). Values between 0 and 1 diminish the contribution of the corresponding component, while values greater than 1 exaggerate them.
Type	Opens a pull-down menu for specification of whether Particle trace calculation uses steady-state data to produce a Streamline or transient data to produce a Pathline.
Stream	Traces a massless Particle in a steady-sate vector field (for steady-state data or the current time-step of transient data).
Path	Traces a massless Particle through a time-varying vector field *and so is only available with transient results data*. On certain systems, this selection can consume significant quantities of CPU time to calculate the resulting Particle trace.
Show As	Opens a dialog for specification of trace representation.

Line	Depicts the trace as a line.
Ribbon	Depicts trace as if it were a ribbon. The ribbon width is a specified fixed value, while the twisting is determined by the rotation of the flow about the path of the trace at any particular point on the trace.
Ribbon Width	This field only applies when Ribbon representation is chosen. Larger values in this field produce wider ribbons.
Emitter Information	
Emitters List	This section shows a list of all emitters created for the currently selected Particle Trace Part.
Emit From	Opens a pull-down menu for the specification of the emitter type.
Cursor	Creates Particle trace beginning from the position of the Cursor tool.
Line	Creates Particle traces beginning from the position of the Line tool.
# Points	This field specifies the number of traces you want to emit from the Line tool.
Plane	Create Particle traces beginning from the position of the Plane tool.
# Points	These fields specify the number of traces you want to emit from the Plane tool in the X and Y axes of the tool.
Part	Creates particle traces beginning from each node of the Part specified by the Part ID Number field.
Part ID Number	This field specifies the Part you wish to use as an emitter for the creation of a particle trace. The Part ID Number for a Part is found in the Main Parts List. (see Section 3.1, Part Overview)
Density	If 1.0, will emit from each node of the part. Lass than 1.0 values indicate a subset of nodes to be used, randomly placed, as emitters.
Interactive Emitter	Toggles on/off interactive Particle tracing. Manipulation of the emitter currently selected in the Emitters List will cause the Particle trace to be recreated at the new location and updated in the Graphics Window. When manipulation of the tool stops, the Particle trace and any Parts that are dependent on it will be updated. (Only available for non-surface-restricted streamlines) (Emitters created by picking a surface or from a Part can not be made interactive).
Add Emit	Adds an emitter of the type specified by Emit From to the currently selected Particle Trace Part.
Delete Emit	Deletes the emitter selected in the Emitters List from the selected Particle Trace Part.
Direction	Trace the Particle in positive time, meaning to trace with the vector field, or trace the Particle in negative time, meaning to trace the Particle upstream. Option only applies to streamlines. Pathlines must be traced in + time.
(+)	Positive time option traces Particle(s) forward in time. (This is the only option for time-dependent datasets.)
(–)	Negative time option traces Particle(s) backward in time.
(+/–)	Positive/Negative time option traces Particle(s) both forward and backward in time.
Total Time Limit	This field specifies the maximum length of time the Particle trace may continue (it may terminate sooner for other reasons).
Emission Time Start	This field specifies the solution time at which to begin Particle emission. Enter value between beginning and ending time available.
Time Delta	This field specifies the time interval between emissions of Particles from the emitters. If "0", only one set of emissions will occur at start time
Surface-Restrict Toggle	Toggles on/off surface restricted feature for streamlines. The streamline will be constrained to stay on the surface of the selected Part(s) by using only the tangential component of velocity. Be sure to use the Pick Surface feature in locating the emitter for a

surface restricted particle trace to ensure that the emitter is located on the surface of a Part.

Pick Surface Toggle	Toggles on/off the feature which allows you to place the trace emitter at a point on a surface directly below the mouse pointer by clicking the left mouse button. This option is forced on if the Surface Restricted Toggle is on.
Variable Offset	This field specifies the distance into the flow field at which velocity (and other variables) are to be sampled for the surface restricted trace(s). If velocity values are present at the surface, this offset can be set to zero.
Display Offset	This field specifies the distance away from the surface to display the surface restricted trace(s). This is provided so z buffer conflicts between surfaces and trace lines can be overcome. A positive distance moves the trace away from the surface in the direction of the surface normal, while a negative distance moves the trace in the opposite direction.
Time Step Determination	Opens a turn-down area for the specification of time-step parameters.
Min Steps	This field is used to specify the minimum number of integration steps to perform in each element.
Min Angle	If angle between two successive line segments of the Particle trace is less than this value EnSight will double the integration step.
Max Angle	If angle between two successive line segments of the Particle trace is greater than this value EnSight will half the integration step.
Rot Angle	If the dot product between successive rotation vectors of the Particle trace is greater than COS(Rot Angle), the integration step is halved.
Massed Particles	Opens a turn-down area for the specification of massed-particle parameters.
Massed Particles	Toggles on/off the massed-particle traces feature. The default is OFF.
Force Terms	Determines which force terms are used in the momentum balance equation calculation.
Drag Term Toggle	Toggles on/off the inclusion of the drag force term in the massed-particle computation. The default is ON.
Gravity Term Toggle	Toggles on/off the inclusion of the gravity force term in the massed-particle computation. The default is ON.
Pressure Term Toggle	Toggles on/off the inclusion of the pressure force term in the massed-particle computation. The default is OFF.
Particle Diameter	This field specifies the diameter of all particles. The default is 1.e-3.
Particle Density	This field specifies the density value of all particles. The default is 1.e+3.
Initial Velocity	Determines what initial velocity to use for all the particle emitters. The default is Use Fluid Toggle ON.
Use Field Toggle	Toggles on/off whether all particle emitters should use the fluid velocity at their corresponding locations. The default is ON.
X, Y, Z	These fields specify the initial velocity components of all particle emitters. Their default is <0., 0., 0.>.
Gravity Vector	These fields specify the gravity vector to be applied in the massed-particle computation. The default gravity components are <0., -9.81, 0.>. This parameter only works with the gravity force term.
Fluid Density	This field specifies the fluid density variable to be used in the massed-particle computation. The default is "None".
Or	This field specifies the fluid density value to be used in the computation if "None" is specified as the variable name. The default value is 1.
Fluid Dynamic	This field specifies the fluid dynamic viscosity variable to be used in the massed-particle

Viscosity	computation. The default is "None".
Or	This field specifies the fluid dynamic viscosity value to be used in the computation if "None" is specified as the variable name. The default value is 1.9620e-5. This parameter only works with the drag force term.
Pressure Gradient	This field specifies the fluid pressure gradient variable to be used in the massed-particle computation. The default is "None". This parameter only works with the pressure force term.
Drag Coefficient Function	This field specifies the drag coefficient function to be called each time the drag coefficient is calculated. This function defaults to "None" which essentially defaults to the table described above. Other functions may be accessed via the User-Defined Math Function facility, i.e. DragCoefTable1(Re) (same as default), DragCoefPoly(Re), DragCoefPower(Re). All functions must take the Reynolds Number as their only argument. This parameter only works with the drag force terms.
Create	(At the bottom of the Feature Detail Editor) Creates the Particle trace Part in the Graphics Window as specified.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How to Create Particle Traces)

Troubleshooting Particle Traces

Problem	Probable Causes	Solutions
Particle Trace Part is empty.	Velocity is zero.	Change time steps or change location of emitters.
	Emitter points are outside of flow field.	Change location for emitter points.
	Dataset is 3D and parent Parts are 2D, or dataset is 2D and parent Parts are not planar.	Change parent Parts.
	The created variable selected does not exist for the parent Part(s)	Recreate the variable for the parent Part(s) selected
Streamline is OK, but pathline is empty.	Creating pathline with the emitter emitting at the last time step.	Modify emitter time for the emitter groups.
Particle trace terminates prematurely	Velocity has gone to zero.	None
	Particle has been traced out of the flow field.	None
	Stopping point is at the boundary between two Parts.	Change the parent Parts for the Particle trace to include neighbor Part.
	Particle getting lost and EnSight's search algorithm failing.	Call CEI hotline support.
	Total Time Limit reached.	Change Total Time Limit.

Problem	Probable Causes	Solutions
Particle trace exists, then is removed after deleting Parts.	The parent Part for the Particle trace was deleted.	None
Particle trace creation requested, but Particles don't come back.	Requested a large number of Particle traces and/or doing pathlines in large transient dataset.	Be patient.
	Particles are stuck in a recirculation area.	Process will finish when Total Time Limit is reached. Consider terminating job and starting over with a smaller Total Time Limit.
Interactive tracing is slow.	The size of the model and density of the mesh will affect the performance of an interactive trace.	If you can, run on a faster, larger memory workstation. Also, limit if possible the area of interest by cutting the mesh into pieces with the Cut & Split Part editing operation.
Interactive trace does not enter the next Part	Interactive tracing is only done through the Part the emitter resides in.	When you let go of the emitter the full trace will be shown
Surface restricted Particle trace does not appear	Zero velocity at chosen variable offset	Select a Variable offset distance that will give nonzero velocity
	Display offset causing trace to be on opposite side of a surface (hidden surface on)	Change sign of the Display offset
	Emitter does not lie on the surface of selected Parts	Create emitters that lie on the surface

7.5   Clip Create/Update

A Clip is a straight line (a Clip Line), a plane (a Clip Plane), a quadric surface (cylinder, sphere, etc.), a constant x, y, or z plane, a box, or an i, j, or k plane that passes through selected model Parts (or already created Clips, Isosurface, or Developed Surface Parts). EnSight calculates the values of variables at the nodes of the Clip. Clips can be parent Parts. For example, you can create a Clip Line passing through a vector field, then create vector arrows originating from the nodes of the Clip Line. Clips are created on the server, and so are not affected by the selected Representation(s) of the parent Part(s). If you activate or create variables after creating a Clip, the Clip automatically updates to include them.

You specify the location, orientation, and size of the Clip numerically in the Transformations Editor dialog, or interactively using the Line, Plane, or Quadric surface tool. If you wish, EnSight will automatically extend the size of a Clip Plane to include all the elements of the parent Part(s) that intersect the plane.

For a Clip Line, which is composed of bar elements, you specify how many evenly spaced nodes are along the line. For a grid-type Clip Plane, which is composed of rectangular elements, you specify the number of nodes in each dimension, resulting in an evenly spaced grid of nodes across the plane.

If you request a mesh-type Clip Plane, an xyz clip, or any of the quadric surfaces, EnSight finds the intersection of the specified plane or surface with the selected parent Part(s) and creates elements of various dimensions, sizes, and shapes that together form a cross-section of the parent Part(s). In this cross-section, three-dimensional parent Part elements result in two-dimensional Clip Plane elements, and two-dimensional parent Part elements result in one-dimensional Clip Plane elements. Note that two-dimensional parent Part elements that are coplanar with the cross-section are not included since they do not intersect the plane.

For XYZ, Plane, Quadric and Revolution Clips you can specify the resulting part to be all elements that intersect the specified value - resulting in a "crinkly" surface which can help analyze mesh quality.

For each Clip node on or inside an element of the selected parent Part(s), EnSight calculates the value of each variable by interpolating from the variable's values at the surrounding nodes of the parent Part(s).

You can interactively manipulate the location of a clip Part by toggling on the Interactive Tool button. When this toggle is on, the tool used to create the clip Part will appear in the Graphics Window. Manipulation of this tool will cause the clip Part to be recreated at the new location. This feature allows you to interactively sweep a plane across your model or manipulate the size and location of the cylinder, sphere, or cone.

You can animate a Clip by specifying an Animation Delta vector that moves the Clip to a new location for each frame or page of the animation. The Clip updates to appear as if it had been newly created at the new location and time.

For structured Parts, you can sweep through the Part with any of the i, j, or k planes.

An XYZ Box Clip will create a subset part which is either inside or outside the specified x,y,z boundaries. The boundaries can be infinite.

Clicking once on the Clip Create/Update Icon opens the Clip Editor in the Quick interaction Area which is used to both create and update clip Parts.
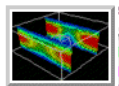

Figure 7-21
Clip Create/Update Icon

Use Tool

IJK   The IJK clip tool is used with structured mesh results.
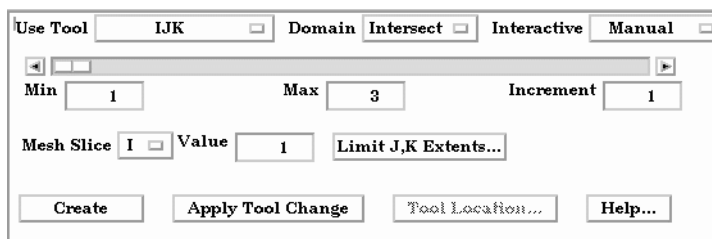

Figure 7-22
Quick Interaction Area - Clip Editor - IJK tool

Domain   Specification to extract the intersection of the specified mesh slice values. For IJK clips, the only valid selection is "Intersect".

Interactive   Opens pull-down menu for selection of type of interactive manipulation of the IJK clip. Options are:

  Off   Interactive IJK clips are turned off.

  Manual   Value of the IJK clip selected are manipulated via the slider bar and the IJK clip is interactively updated in the Graphics Window to the new value.

  Auto   Value of the IJK clip is incremented by the Auto Delta value from the minimum range value to the maximum value. When reaching the maximum it starts again from the minimum.

  Auto Cycle   Value of the IJK clip is incremented by the Auto Increment value from the minimum range value to the maximum value. When reaching the maximum it decrements back to the minimum.

Slider Bar   For IJK clips, the slider bar is used to increment / decrement the Mesh Slice Value between its Minimum and Maximum value.

  Min   Specification of the minimum slice value for the range used with the "Manual" slider bar and the "Auto" and "Auto Cycle" options.

  Max   Specification of the maximum slice value for the range used with the "Manual" slider and the "Auto" and "Auto Cycle" options.

  Increment   Specification of the increment/decrement the slider will move within the min and max, each time the stepper buttons are clicked.

Mesh Slice   Opens a pull-down menu for selecting which of the IJK dimensions you wish to allow to change. You will then specify Min, Max and Step limits for the two remaining "fixed" dimensions.

Value   This field specifies the I, J, or K plane desired for the dimension selected in Mesh Slice

Limit IJK Extents   Opens the "Limits Extents of Current Slice By" dialog, in which the off dimension ranges can be limited.

IJK D(2)Min	This field specifies the minimum value for the second fixed dimension.
IJK D(2)Max	This field specifies the maximum value for the second fixed dimension.
IJK D(2) Step	This field specifies the step size through the second fixed dimension.
IJK D(3)Min	This field specifies the minimum value for the third fixed dimension.
IJK D(3)Max	This field specifies the maximum value for the third fixed dimension.
IJK D(3) Step	This field specifies the step size through the third fixed dimension.
Show Parent IJK Part Extents...	Will show the second and third dimension Min and Max extents as defined for the clip parent Part.
Create	Creates the Clip Part in the Graphics Window as specified.
Apply Tool Change	Recreates the Clip Part selected in the Main Parts List at the current position of and of the type specified by Use Tool.

Feature Detail Editor (Clips) - IJK

Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor, the Creation attributes section of which offers the same features for the IJK tool as the Quick Interaction Area Editor.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How To Create IJK Clips)

Use Tool

XYZ   The XYZ tool is used to create a planar Part at a constant Cartesian component value that is referenced according to the local frame of the part.
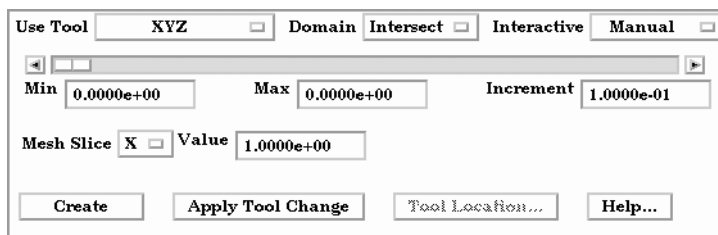
Figure 7-23
Quick Interaction Area - Clip Editor - XYZ Tool

Domain   *Intersect*   will create the cross section of the selected parts at the specified X, Y, or Z plane.

                    Crinkly   will create a new part consisting of the parent part elements that intersect the X, Y, or Z plane

Interactive   Opens pull-down menu for selection of type of interactive manipulation of the XYZ clip. Options are:

  Off   Interactive XYZ clips are turned off.

  Manual   Value of the XYZ clip selected are manipulated via the slider bar and the XYZ clip is interactively updated in the Graphics Window to the new value.

  Auto   Value of the XYZ clip is incremented by the Auto Delta value from the minimum range value to the maximum value. When reaching the maximum it starts again from the minimum.

  Auto Cycle   Value of the XYZ clip is incremented by the Auto Increment value from the minimum range value to the maximum value. When reaching the maximum it decrements back to the minimum.

Slider Bar   For XYZ clips, the slider bar is used to increment / decrement the Mesh Slice Value between its Minimum and Maximum value.

  Min   Specification of the minimum interval value of the interactive XYZ clip.

  Max   Specification of the maximum interval value of the interactive XYZ clip.

  Increment   Specification of the interval step of the interactive XYZ clip.

Mesh Slice   Opens a pulldown menu for selecting which of the XYZ components you wish to clip, i.e. the X, the Y, or the Z component.

Value   This field specifies the coordinate desired for the Mesh Slice component.

Apply Tool Change   Recreates the Clip Part selected in the Main Parts List at the current position of and of the type specified by Use Tool.

Create   Creates the Clip Part in the Graphics Window as specified.

Feature Detail Editor (Clips) - XYZ

Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor (Clips), the Creation attributes section of which offers access to the same interactive clip parameters as found in the Quick Interaction Area Editor, along with additional animation delta control of clips using the XYZ tool.
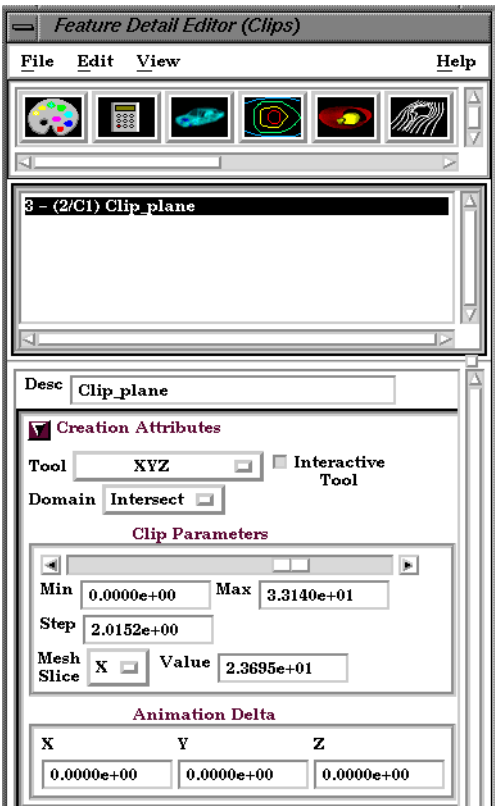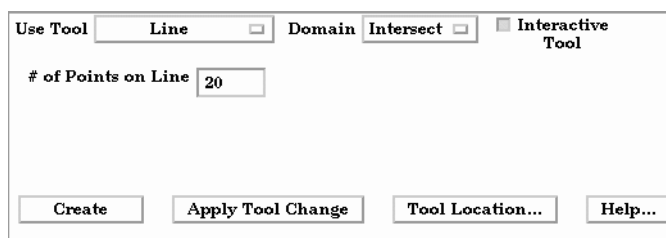

Figure 7-24
Quick Interaction Area - Clip Editor - XYZ Tool - Creation Attributes

Animation Delta

These X,Y,Z fields specify the incremental change in position of the clip for each page of Flipbook or frame of Keyframe animation.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How To Create XYZ Clips)

Use Tool

Line   The Line tool is used to create a clip line.

Figure 7-25
Quick Interaction Area - Clip Editor - Line Tool

Domain   Specification to extract the intersection of the line tool with the selected part(s). For Line
                        clips, the only valid selection is "Intersect".

Interactive Tool   Toggles on/off interactive movement and updating of a clip Part. When toggled on, the
                        line tool used to create the 2D clip line will appear in the Graphics Window. Movement of
                        the tool will cause the Clip Part to be recreated at the new position. When manipulation of
                        the tool stops, the clip Part and any Parts that are dependent on it will be updated. During
                        movement, the Tool itself will not be visible, so as not to obscure the Line Clip Part. The
                        Tool will reappear when the mouse button is released.

# of Points on Line   Specification of number of evenly spaced points on the line at which to create a node.

Tool Location...   Opens the Transformation Editor dialog to permit precise positioning of the Line Tool
                        within the Graphics Window. (see Tool Positions... Line Tool in Section 6.5, Tools Menu
                        Functions and How To Use the Line Tool)

Apply Tool Change   Recreates the Clip Part selected in the Main Parts List at the current position of and of the
                        type specified by Use Tool.

Create   Creates the Clip Part in the Graphics Window as specified.

Feature Detail Editor (Clips) - Line

Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor (Clips), the Creation attributes section of which offers access to additional features for the creation and modification of clips using the Line tool.
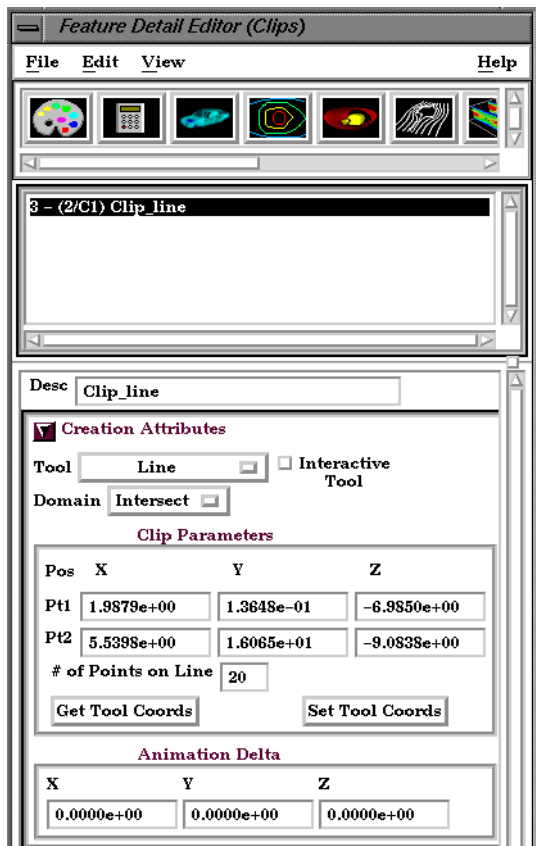
Figure 7-26
Feature Detail Editor (Clips) - Line Tool Creation Attributes

Clip Parameters

Pos of Pt1
Pos of Pt2

Specification of XYZ endpoint-coordinates of Line Clip. The position of a Line Clip Part, if selected in the Feature Detail Editor's Parts List, can be changed by entering values in the numeric fields and then pressing Return.

Set Tool Coords

The position of the Line Clip tool can be changed by entering values in the numeric fields and then pressing Set Tool Coords.

Get Tool Coords

The values in the numeric fields (and the position of a Line Clip Part, if selected in the Feature Detail Editor's Parts List) can be updated after moving the Line tool interactively in the Graphics Window by clicking Get Tool Coords. If a Line Clip Part is selected in the Feature Detail Editor Parts List, it will be repositioned to the new coordinates after clicking Get Tool Coords. Coordinates are always in the original model frame (Frame 0).

Animation Delta

These X,Y,Z fields specify the incremental change in position of the clip for each page of Flipbook or frame of Keyframe animation.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How To Create Line Clips)

Use Tool

Plane The Plane Tool is used to create a Plane Clip.

Domain *Intersect* will create the cross section of the selected parts where they intersect the plane tool.

Crinkly will create a new part consisting of the parent part elements that intersect the plane tool.

Inside will cut the parent parts and create a new part consisting of the portion on the positive z side of the plane tool.

Outside will cut the parent parts and create a new part consisting of the portion on the negative z side of the plane tool.

In/Out will cut the parent parts and create two new parts - namely an *Inside* and *Outside* part.

Plane Type

Mesh Will create a Plane Clip showing the cross section of the parent Part.

Figure 7-27
Quick Interaction Area - Clip Editor - Plane Tool - Mesh Type

Plane Extents Opens a pull down menu for selection of the extent of the Plane Clip.
Finite limits the Plane Clip to the area specified by the Plane Tool corner coordinates.
Infinite extends the Plane Clip to include the intersection of the plane with all elements of the selected model Parts.

Grid Will create a Plane Clip by discrete point sampling.

Figure 7-28
Quick Interaction Area - Clip Editor - Plane Tool - Grid Type

Grid Pts on:XY These fields specify the number of points on each edge of a Plane Clip at which to create nodes. Additional nodes are located in the interior of the plane to form an evenly spaced grid. The values must be positive integers. Applicable only to grid-type Plane Clips. Grid Pts in X correspond to the x-direction on the Plane tool, while the number of Grid Pts in Y correspond to the y-direction of the Plane tool.

Apply Tool Change Recreates the Clip Part selected in the Main Parts List at the current position of and of the type specified by Use Tool.

Interactive Tool Toggles on/off interactive movement and updating of the clip Part. When toggled on, the Plane Tool used to create the clip Part will appear in the Graphics Window. Movement of the Plane Tool will cause the Plane Clip to be recreated at the new position. When manipulation of the tool stops, the clip Part and any Parts that are dependent on it will be updated. During movement, the Tool itself will not be visible, so as not to obscure the Line Clip Part. The Tool will reappear when the mouse button is released.

Tool Location...	Opens the Transformation Editor dialog to permit precise positioning of the Plane Tool. (see Section 6.5, Tools Menu Functions and How To Use the Plane Tool)
Create	Creates the Clip Part in the Graphics Window as specified.
Feature Detail Editor (Clips) - Plane	Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor (Clips), the Creation attributes section of which offers access to additional features for the creation and modification of clips using the Plane tool.
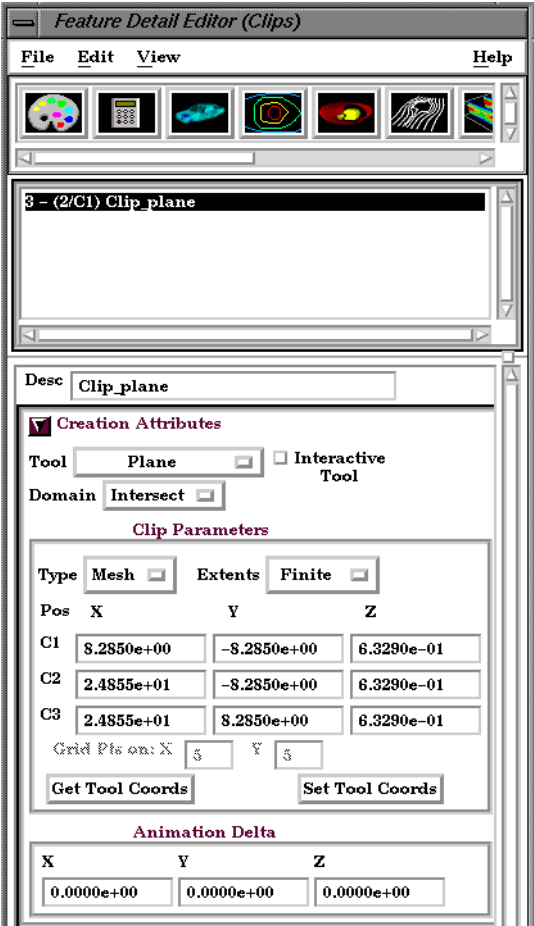
Figure 7-29
Feature Detail Editor (Clips) - Plane Tool Creation Attributes

Clip Parameters

Pos of C1 Pos of C2 Pos of C3	Specification of the location, orientation, and size of the Plane Clip using the coordinates (in the Parts reference frame) of three corner points, as follows: Corner 1 is corner located in negative-X negative-Y quadrant Corner 2 is corner located in positive-X negative-Y quadrant Corner 3 is corner located in positive-X positive-Y quadrant
Set Tool Coords	Will reposition the Plane Tool to the position specified in C1, C2, and C3.
Get Tool Coords	Will update the C1, C2, and C3 fields to reflect the current position of the Plane Tool.
Animation Delta	These X,Y,Z fields specify the incremental change in position of the clip for each page of Flipbook or frame of Keyframe animation.
	(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),
	(see How To Create Plane Clips)

Use Tool

Cylinder, Sphere, Cone These Tools are used to create a quadric clip surface

Domain	*Intersect*	will create the cross section of the selected parts where they intersect the quadric tool.
	Crinkly	will create a new part consisting of the parent part elements that intersect the quadric tool.
	Inside	will cut the parent parts and create a new part consisting of the portion on the inside of the quadric tool.
	Outside	will cut the parent parts and create a new part consisting of the portion on the outside of the quadric tool.
	In/Out	will cut the parent parts and create two new parts - namely an *Inside* and *Outside* part.

.
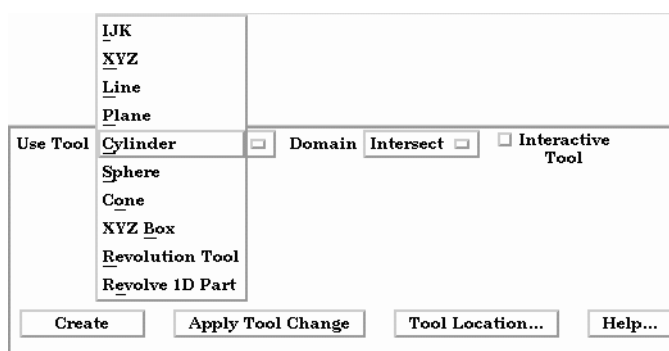
Figure 7-30
Quick Interaction Area - Clip Editor - Cylinder, Sphere, & Cone Tools

Interactive Tool   Toggles on/off interactive movement and updating of a clip Part. When toggled on, the Quadric Tool used to create the Clip Part will appear in the Graphics Window at the location of the Clip Part. Movement of the Quadric Tool will cause the Clip Part to be recreated at the new position. When manipulation of the tool stops, the Clip Part and any Parts that are dependent on it will be updated. During movement, the Tool itself will not be visible, so as not to obscure the Line Clip Part. The Tool will reappear when the mouse button is released.

Tool Location...   Opens the Transformation Editor dialog to permit precise positioning of Quadric Tools.(see Section 6.5, Tools Menu Functions and How To Use the Cylinder Tool, How To Use the Sphere Tool, and How To Use the Cone Tool)

Apply Tool Change   Recreates the Clip Part selected in the Main Parts List at the current position of and of the type specified by Use Tool.

Create   Creates the Clip Part in the Graphics Window as specified.

Feature Detail Editor (Clips) Quadric Tool

Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor (Clips), the Creation attributes section of which offers access to additional features for the creation and modification of clips using the Quadric tools.
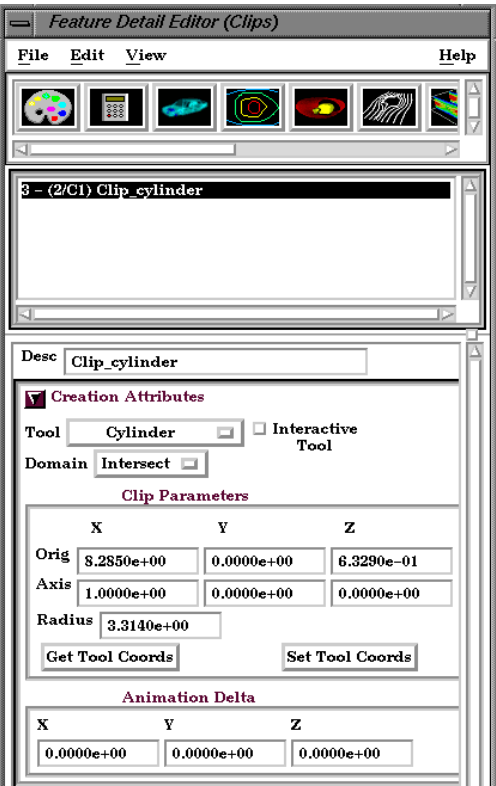

Figure 7-31
Feature Detail Editor (Clips) - Quadric Tool Creation Attributes

Clip Parameters

Cylinder

Orig XYZ	Specification of the origin (the center point) of the Cylindrical Clip.
Axis	Specification of the axis direction of the Cylindrical Clip.
Radius	Specification of the radius of the Cylindrical Clip.

Sphere

Orig	Specification of the origin (the center point) of the Spherical Clip.
Axis	Specification of the axis direction of the Spherical Clip.
Radius	Specification of the radius of the Spherical Clip.

Cone

Orig	Specification of the origin (the center point) of the Conical Clip.
Axis	Specification of the axis direction of the Conical Clip.
Angle	Specification of the angle of the Conical Clip.

Set Tool Coords — Will reposition the Quadric Tool to the position specified in the Clip Parameter fields.

Get Tool Coords — Will update the Clip Parameter fields to reflect the current position of the Quadric Tool.

Animation Delta — These X,Y,Z fields specify the incremental change in position of the clip for each page of Flipbook or frame of Keyframe animation.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics

Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How To Create Quadric Clips)

Use Tool

XYZ Box   This Clipping Tool extracts all elements that are entirely inside of a specified box, or its complement

Domain   *Inside*   will extract the elements of the parent parts that lie entirely within the box.

                Outside   will extract the elements of the parent parts that do not lie entirely within the box.

                In/Out   will create two new parts - namely the *Inside* and *Outside* parts.

.

Figure 7-32
Quick Interaction Area - Clip Editor - Cylinder, Sphere, & Cone Tools

Infinite   The Min and Max bounding for x, y, and z can be set to infinity (or negative infinity in the case of Min).

Min   Specify the minimum box coordinates for x, y, and z (used only when respective Infinity toggles are off).

Max   Specify the maximum box coordinates for x, y, and z (used only when respective Infinity toggles are off).

Apply Tool Change   Recreates the Clip Part selected in the Main Parts List at the current position of and of the type specified by Use Tool.

Create   Creates the Clip Part in the Graphics Window as specified.

Feature Detail Editor (Clips) - XYZ Box   Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor, the Creation attributes section of which offers the same features for the XYZ Box tool as the Quick Interaction Area Editor.

                (see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

                (see How To Create XYZ Box Clips)

Use Tool

Revolution Tool   This clipping Tool is used to create custom clip surfaces which are defined by revolving a set of lines about a defined axis.
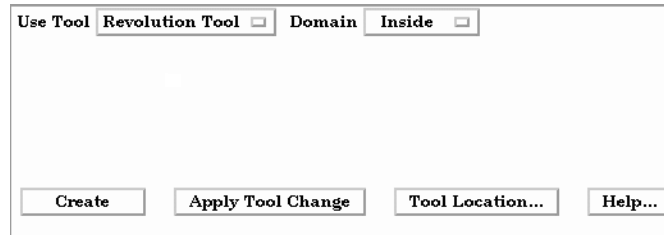
Figure 7-33
Quick Interaction Area - Clip Editor - Revolution Tool

Domain

Intersect   will create the cross section of the selected parts where they intersect the plane tool.

Crinkly   will create a new part consisting of the parent part elements that intersect the plane tool.

Inside   will cut the parent parts and create a new part consisting of the portion on the positive z side of the plane tool.

Outside   will cut the parent parts and create a new part consisting of the portion on the negative z side of the plane tool.

In/Out   will cut the parent parts and create two new parts - namely an *Inside* and *Outside* part.

Tool Location...   Opens the Transformation Editor dialog to permit precise location of the revolution tool within the Graphics Window. It is here where you also can control the number and positioning of the set of lines which make up the tool.
(see Section 6.5, Tools Menu Functions and How To Use the Surface of Revolution Tool)

Apply Tool Change   Recreates the Clip Part selected in the Main Parts List at the current position of and of the type specified by Use Tool.

Create   Creates the Clip Part in the Graphics Window as specified.

| Feature Detail Editor (Clips) - Revolution Tool | Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor (Clips), the Creation attributes section of which offers access to additional features for the creation and modification of clips using the Revolution tool. |
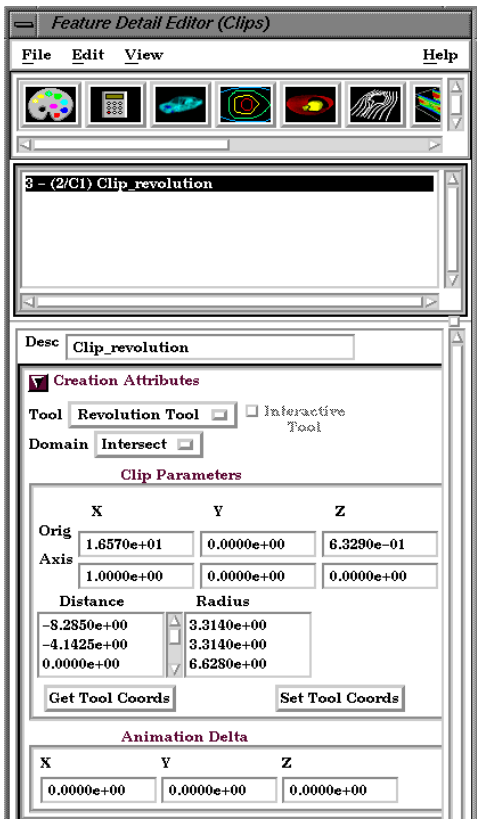
Figure 7-34
Feature Detail Editor (Clips) - Revolution Tool Creation Attributes

Revolution Tool Clip Parameters

Orig	These fields specify the XYZ coordinates of the origin (center point) of the Revolution Clip.
Axis	These fields specify the XYZ coordinates of the axis direction of the Revolution Clip.
Distance/Radius	These lists specify the distance (from the origin) and radius for each point that defines the Revolution Clip. The points can Not be edited within this dialog. You must edit the Revolution Tool in the Transformations dialog.
Set Tool Coords	Will reposition the Revolution Tool to the position specified in the Clip Parameter fields.
Get Tool Coords	Will update the Clip Parameter fields to reflect the current position of the Revolution Tool.
Animation Delta	These X,Y,Z fields specify the incremental change in position of the clip for each page of Flipbook or frame of Keyframe animation.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see Section 6.5, Tools Menu Functions and How To Use the Surface of Revolution Tool)

Use Tool

Revolve 1D Part   This option will create a clip surface by revolving a line, defined by a Part, about an axis.

Figure 7-35

Quick Interaction Area - Revolve 1D Part Clip Editor

Domain

Intersect   will create the cross section of the selected parts where they intersect the plane tool.

Crinkly   will create a new part consisting of the parent part elements that intersect the plane tool.

Inside   will cut the parent parts and create a new part consisting of the portion on the positive z side of the plane tool.

Outside   will cut the parent parts and create a new part consisting of the portion on the negative z side of the plane tool.

In/Out   will cut the parent parts and create two new parts - namely an *Inside* and *Outside* part.

Revolve Part   This field specifies the Part number which will be revolved. The Part must contain only bar elements and must have only two free ends (i.e., there must be only one "logical" line contained in the Part).

Orig   These fields specify the XYZ coordinates of the axis line origin point.

Axis   These fields specify the direction vector of the axis line. The "line" contained in the Part specified by number in Revolve Part will be revolved about this axis to create the clip surface Part.

Apply Tool Change   Recreates the Clip Part selected in the Main Parts List at the current position of and of the type specified by Use Tool.

Create   Creates the Clip Part in the Graphics Window as specified.

General Quadric

Feature Detail Editor (Clips)

Double Clicking on the Clip Create/Update Icon brings up the Feature Detail Editor (Clips), the Creation attributes section of which offers access to offers access to one type of clip creation which is not available in the Quick Interaction area. It is possible to create a 3D Quadric clip using the General Quadric option by directly specifying the coefficients of a general quadric equation.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),
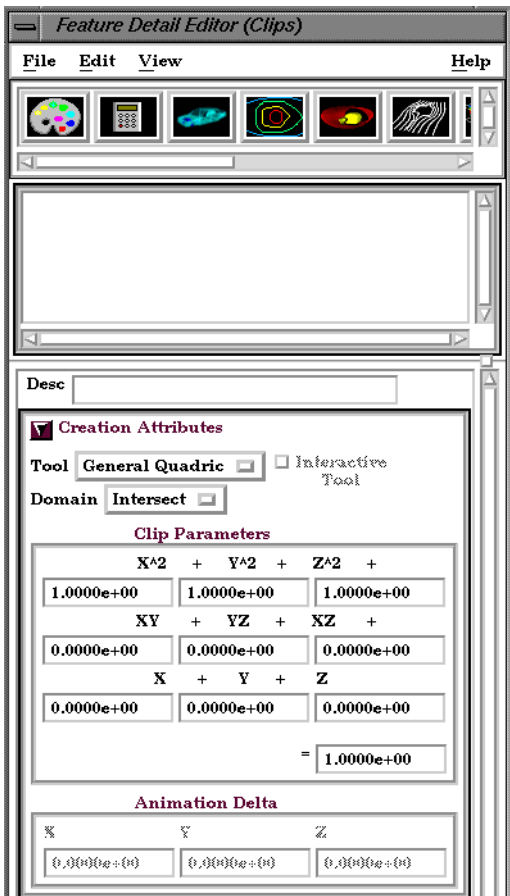

Figure 7-36
Feature Detail Editor (Clips) - Revolve 1D Part Creation Attributes

10 coefficient values

These coefficient values represent the general equation of a Quadric surface. They can be changed by modifying the values. No tool exists corresponding to this equation.

$$AX^2+BY^2+CZ^2+DXY+EYZ+FXZ+GX+HY+IZ=J$$

Animation Delta

Not available for General Quadric Clips.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics

Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

Troubleshooting Clips

Problem	Probable Causes	Solutions
Clip does not move during animation	Animation deltas are not set, or are too small.	Change the animation delta values.
Clip results in an empty Part.	Clip was taken outside of the model.	Change the clip Tool location.

7.6 Vector Arrow Create/Update

Vector Arrows visualize the magnitude and direction of a vector variable at discrete points (at nodes, element vertices, or at the center of elements).

Other features can visualize magnitude, but Vector Arrows also show direction.
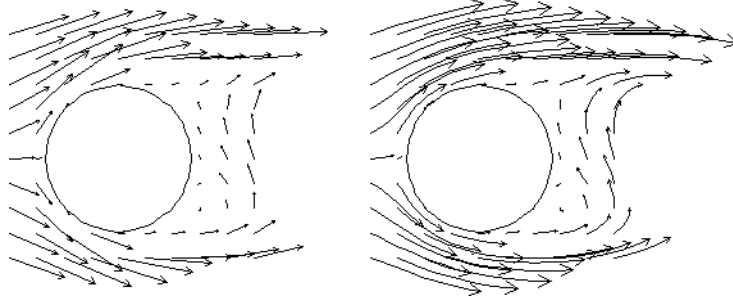
Vector arrow Parts are dependent Parts known only to the client. They cannot be used as a parent Part for other Part types and cannot be used in queries. As dependent Parts, they are updated anytime the parent Part and/or the creation vector variable changes (unless the general attribute Active flag is off).

Vector arrows can be filtered according to low and/or high threshold values.

Vector arrows can emanate from the available nodes of the parent Part(s), the available element vertex nodes of the parent Part(s), or the available element centers of the parent Part(s) which pass through the filter successfully. The nodes and elements available in the parent Part are based on the visual Representation of the Part. Thus, for a border Representation of a Part, only the border elements and associated nodes are candidates.

Vector arrows can have straight shafts representing the vector at the originating location, or be the segment of a streamline emanating from the originating location (curved). Straight vector arrows are displayed relatively quickly, while curved vector arrows can be time consuming.

Different tip styles, sizes, and colors can be used to enhance vector arrow display.

Clicking once on the Vector Arrow Create/Update Icon opens the Vector Arrow Editor section of the Quick interaction Area which is used to both create and update (make changes to) vector arrow Parts.

Figure 7-37
Vector Arrow Create/Update Icon

Figure 7-38
Quick Interaction Area - Vector Arrow Editor

Scale Factor / Time	When Type is "Rectilian", this field specifies a scale factor to apply to the vector values before displaying them. Scaling is usually necessary to control the visual length of the vector arrows since the vector values may not relate well to the geometric dimensions. Can be negative, causing the vector arrows to reverse direction.
	When Type is "Rect. Fixed", this field specifies the length of the arrows in units of the model coordinate system. Can be negative, causing the vector arrows to reverse direction.
	When Type is "Curved", this field specifies the duration time for streamlines forming the shaft of curved vector arrows. Is an indication of the length of the curved vector arrow.
Get Default	Sets Scale or Time Factor value to a computed reasonable value based on the vector variable values and the geometry.
Arrow Tips...	Opens the Vector Arrow Tip Settings dialog.
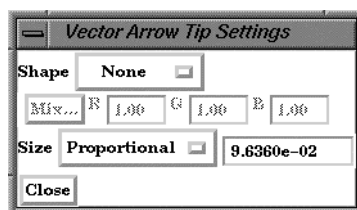
Figure 7-39
Vector Arrow Tip Settings dialog

Shape	Opens a pop-up menu to select tip shape.
	None option displays arrows as lines without tips.
	Normal arrows have two short line tips, similar to the way many people draw arrows by hand. The tip will lie in the X–Y, X–Z, or Y–Z plane depending on the relative magnitudes of the X, Y, and Z components of each individual vector. Suggested for 2D problems.
	Triangles arrows have a tip composed of two intersecting triangles in the two dominant planes. Good for both 2D and 3D fields.
	Tipped arrows display the tip of the arrow in any user specified color. Good for both 2D and 3D fields. The color may be specified in the RGB fields or chosen from the Color Selector dialog which is opened by pressing the Mix... button
Size	Opens a pop-up menu for selecting tip size.
	Fixed sized arrows have tips for which the length is specified in the data entry field to the right of the pop-up menu button. Units are in the model coordinate system.
	Proportional sized arrow tips change proportionally to the change in the magnitude of the vector arrows.
Type	Opens a pop-up menu for selection of shaft-type of vector arrows. Options are:
	Rectilian arrows have straight shafts. The arrow points in the direction of the vector at the originating location. The length of the arrow shaft is determined by multiplying the vector magnitude by the scale factor.
	Rect. Fixed arrows have straight shafts. The arrow points in the direction of the vector at the originating location. The length of the arrow shaft is determined by the scale factor. It is independent of the vector variable.
	Curved arrows have curved shafts. The arrow is actually a streamline emanating from the originating location. It represents the path that a massless Particle would follow if the flow field was steady state. For this option, the "Scale Factor" changes to "Time". Time is the amount of time the streamline is allowed to take and is an indication of how long the arrow will be. Hint: Since curved arrows can take a significant amount of time(depending on the number of originating locations), the setting of a proper "Time" value is

critical. The best way to do this is to first do a single Particle trace at a representative location with the estimated "Time" value as the Max Time. A quick iteration or two on the value here could save considerable time for the curved vector arrow computation.

Location

Opens a pop-up dialog for the selection of root-location of arrow shafts. The options are:

Node arrows originate from each node of the parent Part(s).

Note: Discrete Particles Parts must use Node option.

Vertices arrows originate only from those nodes at the vertices of each element of the parent Part(s) (i.e., arrows are not displayed at free nodes or mid-side nodes).

Element Center arrows originate from the geometric center of each element of the parent Part(s).

Display Offset

This field specifies the distance away from a surface to display the vector arrows (so that potential Z-buffer conflicts can be overcome). A positive distance value moves the vector arrows away from the surface in the direction of the surface normal.

Density

The fraction of the parent's nodes/elements which will show a vector arrow. A value of 1.0 will result in a vector arrow at each node/element, while a value of 0.0 will result in no arrows. If between these two values, the arrows will be distributed randomly at the specified density.

Filter

Selection of pattern for filtering Vector Arrows according to magnitude. Options are:

None displays all the vector arrows. No filtering done.

Low displays only those arrows with magnitude above that specified in the Low field. Filters low values out.

Band displays only those arrows with magnitude below that specified in the Low field and above that specified in the High field. Filters the band out.

High displays only those arrows with magnitude below that specified in the High field. Filters the high values out.

Low_High displays only those arrows with magnitude between that specified in the Low field and that specified in the High field. Filters out low and high values.

Apply New Variable

Changes the vector Variable used to create the Vector Arrows to that currently selected in the Variables List.

Feature Detail Editor (Vector Arrows)

Double clicking on the Vector Arrow Create/Update Icon opens the Feature Detail Editor for Vector Arrows, the Creation Attributes Section of which provides access to the functions available in the Quick Interaction Area plus two more:

Figure 7-40
Feature Detail Editor (Vector Arrows)

Projection

Opens a pop-up menu to allow selection of which vector components to include when calculating both the direction and magnitude of the vector arrows. The vector components at the originating point are always first multiplied by the Projection Components (see below). Then one of the following options is applied:

All, to display a vector arrow composed of the Projection-Component-modified X, Y, and Z components.

Normal, to display a vector which is the projection of the All vector in the direction of the normal at the originating location.

Tangential, to display a vector which is the projection of the All vector into the tangential plane at the originating location.

Component, to display both the Normal and the Tangential vectors

The *All, Normal*, and *Tangential* options produce a single vector per location, while the *Component* option produces two vectors per location. If selection is not applicable to a Particular element, that element's vector arrow uses the *All* projection.

Projection Components *X Y Z* These fields specify a scaling factor for each coordinate component of each vector arrow used in calculating both the magnitude and direction of the vector arrow. Specify 1 to use the full value of a component. Specify 0 to ignore the corresponding vector component (and thus confine all the vector arrows to planes perpendicular to that axis). Values between 0 and 1 diminish the contribution of the corresponding component, while values greater than 1 exaggerate them. Negative values reverse the direction of the component. Always applied before the Projection options above.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How to Create Vector Arrows)

Troubleshooting Vector Arrows

Problem	Probable Causes	Solutions
Vector arrows do not match up with their originating locations on one or more of the parent Parts.	Displacements are on for some of the parent Parts, but not others. Or the parent Parts have been assigned to different coordinate frames	Create separate vector arrow Parts for the parents that will be displaced (or assigned to different frame) and the ones that will not be displaced (or assigned to different frames).
You are displaying several different vector arrow Parts at once and can't tell which is which.	Just too much similar information in the same area.	Use different attributes for the different vector arrow Parts, or better yet, display the conflicting vector arrow Parts on separate Part copies which have been moved apart.
You are trying to display vector arrows on a Discrete Particle Part, but can't get them to show up	Arrow Location set to Vertices (the default).	Set the Arrow Location to Nodes.
	No vector data provided for the Discrete Particle dataset, thus values all set to zero when read into EnSight.	Provide vector data for the particles. Specify in the Measured results file. See Section 3.7.

7.7 Elevated Surface Create/Update

Elevated Surfaces visualize the value of a variable by creating a surface projected away from the 2D elements of the parent Part. It is easiest to describe this feature if you think of a planar Part as the parent Part. Now warp this surface up out of plane proportionally to the value of a variable. The resultant surface is an Elevated Surface. Elevated surfaces are to surfaces what Profiles are to lines. While planar surfaces are perhaps the most useful parent Parts to use, parents do not have to be planar. Model Parts containing 2D elements, Clip Planes, Isosurfaces, and even other elevated surfaces are all valid parent Parts.
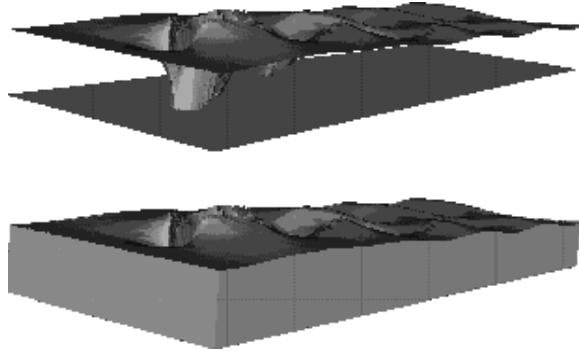
Figure 7-41
Elevated Surface example, with and without Sidewalls

The parent Part is not actually changed, a new surface is created. As this new surface is "raised", projection (Sidewall) elements can be created stretching from the parent to the elevated surface around the boundary of the surfaces if desired. Just the surface, just the sidewalls, or both can be created.

The projection from a node on the parent Part will be in the direction of the normal at the node. If the node is shared by multiple elements, the average normal is used.

The projected distance from a parent Part's node to the corresponding elevated surface node is calculated by adding to the variable's value an Offset value, then multiplying the sum by a Scaling value. Adding the Offset enables you to shift the zero location of the plane. An Offset performs a "shift", but does not change the "shape" of the resulting elevated surface. The Scaling factor changes the distance between parent and elevated surface, a "stretching" effect. EnSight will provide default values for both factors based on the variable's values at the parent Part's nodes.

Clicking once on the Elevated Surface Create/Update Icon opens the Elevated Surface Editor in the Quick Interaction Area which is used to both create and update (make changes to) elevated surface Parts.
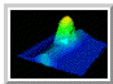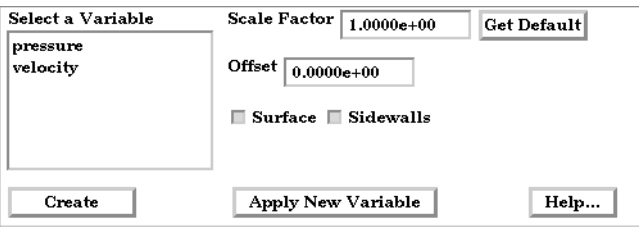

Figure 7-42
Elevated Surface Create/Update Icon

Figure 7-43
Quick Interaction Area - Elevated Surface Editor

Scale Factor	This field specifies the scaling for magnitude of distance between the parent Part node and the corresponding elevated surface node. The Factor is multiplied times the value of the variable. Values larger than one increase the size and values smaller than one decease the size. A negative value will have the effect of switching the direction of the projected surface.
Get Default	Click to set Scale Factor and Offset values to the calculated defaults based on the variable values for the parent Part.
Offset	Value specified is added to the variable values before the Scale Factor is applied to change the magnitude of projected distance. Default offset is magnitude of most-negative projection distance (will cause the surface to be projected positively). Has the effect of shifting the surface plot, but does not change the surface plot shape.
Surface Toggle	Toggles on/off the creation of the actual elevated surface. The sidewalls alone will be created if this toggle is off.
Sidewalls Toggle	Toggles on/off the creation of the sidewalls of the Elevated Surface. Elements will stretch from the parent Part to the Elevated surface around the boundary of the surfaces. The Elevated Surface alone will be created if this toggle is off.
Apply New Variable	Changes the variable the Elevated Surface Part is based on to that currently selected in the Variables List.

Feature Detail Editor
(Elevated Surfaces)

Double clicking on the Elevated Surfaces Create/Update Icon opens the Feature Detail Editor for Elevated Surfaces, the Creation Attributes Section of which provides access to all of the functions available in the Quick Interaction Area plus one more:
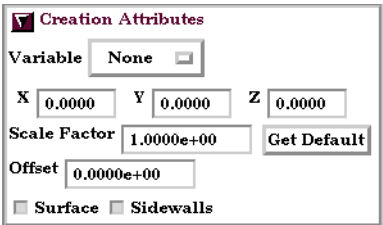

Figure 7-44
Feature Detail Editor (Elevated Surfaces)

X Y Z

For vector-based or coordinate-based elevated surfaces, specify vector components used in creating the elevated surface. Not applicable to scalar-type elevated surfaces. Are according to the reference frame of the Elevated Surface-Part. Letters labeling dialog data entry fields depend on type of the reference frame (Rectangular, Spherical, or Cylindrical). If all components are 0.0, the vector or coordinate magnitude will be used.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How to Create Elevated Surfaces)

Troubleshooting Elevated Surfaces

Problem	Probable Causes	Solutions
The entire Elevated Surface is not projected in the direction you want.		Change the sign of the scale factor.
You have a non-planar parent Part and the elevated surface seems to have strange intersecting elements.	Sidewall elements are not appropriate	Turn off sidewall toggle.
	Scale factor too large.	Lower the Scale Factor.
The Elevated Surface projection appears to be "confused" at various locations.	Inconsistently ordered elements, such that the normals are not "consistent"	Modify element ordering to be consistent, if possible.

7.8 Profile Create/Update

Profiles visualize values of a variable along a line with a plot projecting away from the line. Projectors are parallel to a plane, but not necessarily in a plane. Hence, Profile can follow the line.

You can scale and offset projectors. The positive direction is set with the center point of the Plane Tool (away from center point is positive). Consider a base-line (not necessarily straight) along which the value of a variable is known. Moving along this base-line, you can "plot" the value of the variable on an "axis" whose origin moves along the base-line and whose orientation varies so that it is always both perpendicular to the base-line and parallel to a specified *plane* (but not necessarily parallel to a *line*, enabling the plot-line to follow the curve of the base-line in one dimension). A surface joining the base-line to the plot-line is called a *profile*.
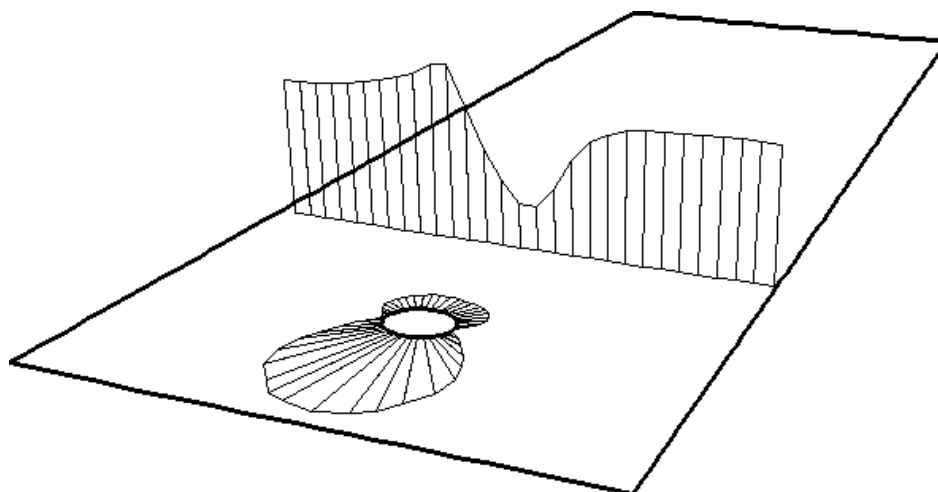
The parent Part of a Profile-Part can be a 2D-Clip Line, a Contour, a Particle Trace, or a model Part consisting of a chain of bar elements. From each node of the parent Part, EnSight draws a "projector" whose length is proportional to the value of the variable at the node, and whose orientation makes it (1) parallel to a specified plane, (2) pointing in a direction corresponding to the sign of the variable's value at the node (with the negative-direction determined by the location of a specified point), and (3) perpendicular to the base-line elements adjoining the node, or, if the base-line bends at the node, oriented so that its projection into the plane defined by the base-line elements will bisect the angle formed by the base-line elements. The outer-end of each projector is connected to those of its neighbors, forming a series of four-sided polygons and hence a surface.

The appearance of the profile depends greatly on the position of the specified sign-direction point (From Point) and the orientation of the specified plane, which you can specify numerically or with the Plane tool. EnSight calculates the projectors using the vector cross-product of the specified-plane's normal (the Z-axis) and each parent Part element, thus you should orient the plane so that its normal is not parallel to the parent Part elements.

The projector length is calculated by adding to the variable's value an Offset value, then multiplying the sum by a Scaling value. Adding the Offset enables you to shift the zero location of the projectors, which might be useful if you wanted to make all the projectors have the same sign. An offset performs a "shift", but does not change the "shape" of the resulting profile. The Scaling factor changes the displayed size of the profile, a "stretching" type of action.   EnSight will provide default values for both factors based on the variable's values at the parent Part's nodes.

Clicking once on the Profile Create/Update Icon opens the Profile Editor the Quick Interaction Area which is used to both create and update (make changes to) profile Parts.
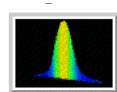
Figure 7-45
Profile Create/Update Icon

Figure 7-46
Quick Interaction Area - Profile Editor

Scale Factor	This field specifies the scaling for magnitude of the projector. The Scale Factor is multiplied times the value of the variable. Values larger than one increase the size and values smaller than one decrease the size.
Offset	The value specified in this field is added to the variable values before the Scale Factor is applied to change the magnitude of projectors. Default offset is magnitude of most-negative projector (making them all positive). Has the effect of shifting the plot, but does not change the plot shape.
Get Default	Click to set Scale Factor and Offset values to the calculated defaults based on the variable values for the parent Part.
Show Orientation Tool	Causes the Plane Tool to become visible in the Graphics Window at the location specified
Update Orientation	Recreates the Profile Part at the current location and orientation of the Plane Tool.
Apply New Variable	Changes the variable the Profile Part is based on to that currently selected in the Variables List.

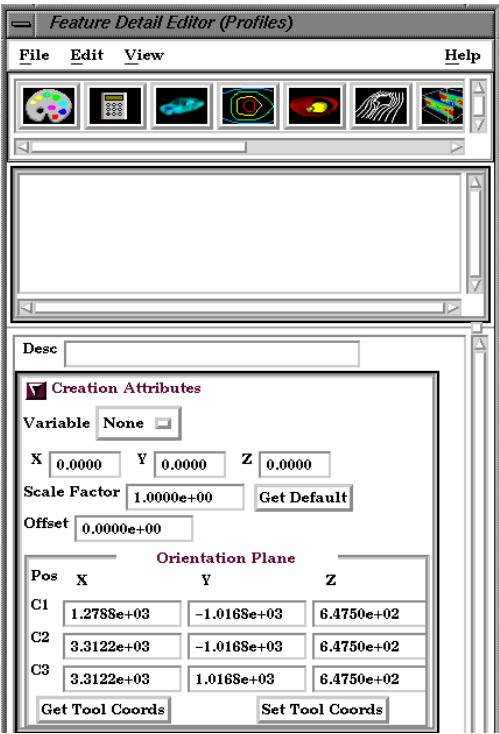|	
Feature Detail Editor (Profiles)	Double clicking on the Profile Create/Update Icon opens the Feature Detail Editor for Profiles, the Creation Attributes Section of which provides access to additional functions for the creation and modification of Profiles:

Figure 7-47
Feature Detail Editor (Profiles)

X Y Z	These fields specify the vector components used in creating the Part for vector based or coordinate-based Profiles. These fields are not applicable to Scalar-based Profiles. When all fields are zero, the magnitude of the Variable value is used. If a value other than zero is entered into a field, the sum of $(Vector_X*X)+(Vector_Y*Y)+(Vector_Z*Z)$ is used as the variable value.

Orientation Plane
  Pos of C1
  Pos of C2
  Pos of C3
	Specification of the location, orientation, and size of the Plane Clip using the coordinates (in the Parts reference frame) of three corner points, as follows:
	      Corner 1 is corner located in negative-X negative-Y quadrant
	      Corner 2 is corner located in positive-X negative-Y quadrant
	      Corner 3 is corner located in positive-X positive-Y quadrant

Set Tool Coords	Will reposition the Plane Tool to the position specified in C1, C2, and C3.

Get Tool Coords	Will update the C1, C2, and C3 fields to reflect the current position of the Plane Tool.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How To Create Profile Plots)

Troubleshooting Profiles

Problem	Probable Causes	Solutions
The entire profile is not projected the direction you want.	The Plane is not oriented correctly.	Turn on the Plane tool so you can see its orientation. The projectors will be parallel to this plane, so adjust its orientation.
	The From Point is not in a good location	Turn on the Plane tool so you can see the location of the center of the plane. Positive projectors will go away from this point, negative towards.
Portions of the profile appear to be projected in the wrong direction.	The From Point is not in a good location.	Turn on the Plane tool so you can see the location of the center of the plane. Positive projectors will go away from this point, negative towards.
	The normal to the Plane is parallel to some of the elements of the parent Part.	Turn on the Plane tool so you can see its orientation. Try to make sure the Z axis of the Plane tool does not lie parallel to any portions of the parent Part.
	The Parent Part does not contain elements which are consistently ordered	None

7.9   Developed Surface Create/Update

A Developed Surface is generated by treating any 2D Part (or parent Part) as a surface of revolution, and mapping specific curvilinear coordinates of the revolved surface into a planar representation.

A Developed Surface derives its name from the implied process that defines a developable surface. A surface is considered "developable" if it can be unrolled onto a plane without distortion. Although every 2D Part in EnSight is not by definition a developable surface, each 2D Part can nevertheless be developed into a planar surface which is distorted according to the type of developed projection specified. For example, a Cylinder Clip Part is by definition a developable surface, because it can be developed into planar surface without distortion. Whereas, a Sphere Clip Part is not a developable surface, because it can not be developed into a planar surface without distortion.

Parent Parts   Only 2D Parts are developed. Also, only one Part is developed at a time. While all 2D Parts qualify as candidate parent Parts, only 2D Parts of revolution are developed coherently. The current developed surface algorithm treats all parent Parts as surfaces of revolution that are developed according to a local origin and axis of revolution. These attributes are either inherited from the parent Part, or must be specified according to the parent Part.

A developed surface permanently inherits the local origin and axis of revolution information from any parent Part created via the cylinder, cone, sphere, or revolution Clip tools. Whereas, surfaces developed from non-Clip Parts require this information to be specified via the Orig. and Axis fields in the Attributes (Developed Surfaces) dialog. The latter case is the only time the values in these fields are used. Although default values are provided, it is up to you to make sure that valid values are specified. In the former case, the Orig and Axis fields only provide convenient feedback of the selected Clip Part. Note that developed surfaces resulting from parent Parts of revolution created via the general quadric Clip tool do not inherit the local origin and axis of revolution attributes from the General Quadric Clip parent; rather, these attributes must be specified.

Figure 7-48
Developed
Surface
Examples
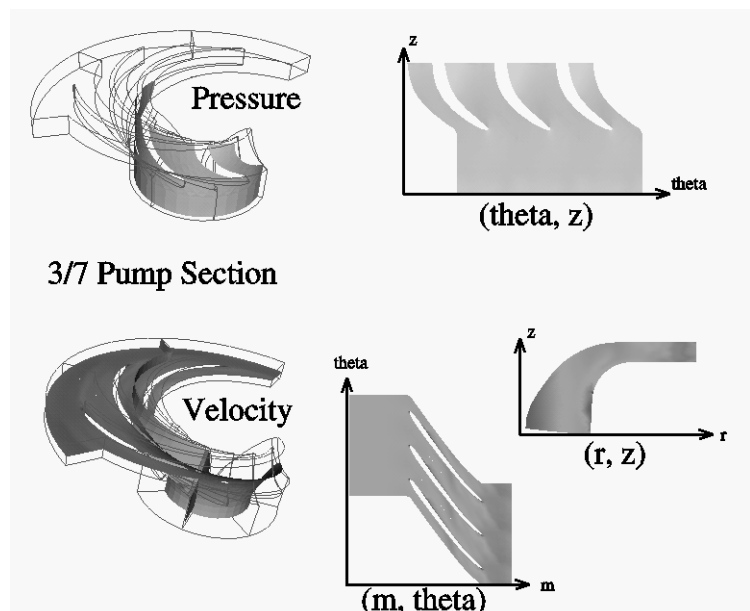
Developed Projections A parent Part is developed by specifying one of three curvilinear mappings called *developed projections*; namely, an (r,z), (θ,z), or (m,θ) projection. The curvilinear coordinates r, θ, z, and m stand for the respective radius, θ, z, and meridian (or longitude) directional components which are defined relative to the local origin and axis of revolution of the parent Part. The meridian component is defined as m = SQRT($r^2 + z^2$).
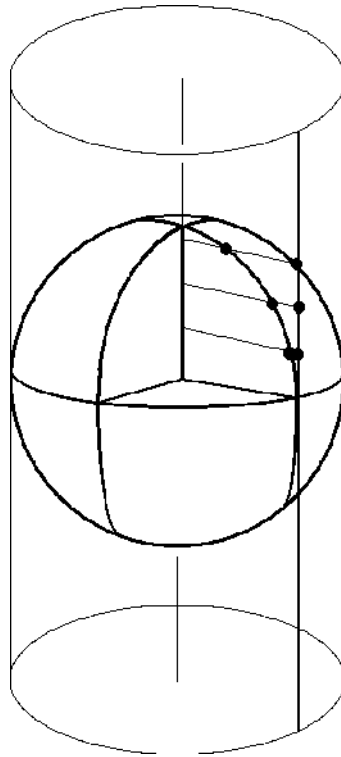
Essentially, each topological projection first surrounds the parent Part of revolution with a virtual cylinder of constant radius. The curvilinear coordinates of the parent Part are then projected along the normals of (and thus onto) the virtual cylinder. Finally, the virtual cylinder is slit along a straight line, or generator, and unwrapped into a plane. This process yields an *equiareal,* or *area preserving*, mapping which means that the area of any enclosed curve on the surface of the parent Part is equal to the area enclosed by the image of the enclosed curve on the developed plane. Although equiareal mappings provide reduced shape distortion, they do suffer from angular distortions of local scale.

Vector fields of the parent Part (for all three developed projections) are developed such that a vector's angle to its surface normal is preserved. For example, a vector normal to the parent surface remains normal when developed onto the planar surface.

Figure 7-49
Developed Equiareal Projection

Seam Line A surface of revolution is developed about its axis, starting at its "seam" line (or zero meridian) where the surface is to be slit. Surface points along the seam are duplicated on both ends of the developed Part. The seam line is specified via a vector that is perpendicular to and originates from the axis of revolution, and which points toward the seam which is located on the surface at a constant value. This vector can be specified either manually or interactively. Interactive seam line display and manipulation is provided via a slider in the Attributes (Developed Surfaces) dialog.

Clicking once on the Developed Surface Create/Update Icon opens the Developed Surface Editor in the Quick Interaction Area which is used to both create and update (make changes to) developed surface Parts.
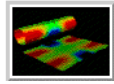

Figure 7-50
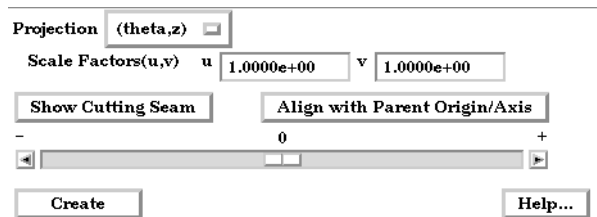Developed Surface Create/Update Icon

Figure 7-51
Quick Interaction Area - Developed Surface Editor

Projection   Opens a pop-up dialog for the specification of which type of (u,v) projection, or mapping, you wish to use for developing a surface of revolution; where u,v denotes curvilinear components of the parent Part that are mapped into the xy-plane of reference Frame 0. The options are:

(r,z)   denotes the radial and z-directional components of the revolved surface

(theta,z)   denotes the theta and z-directional components of the revolved surface.

(m,theta)   denotes the meridian and theta components of the revolved surface. The meridian component is the curvilinear component along a revolved surface that runs in the direction of its axis of revolution (e.g. the meridonal and z-directional components along a right cylinder are coincident, and for a sphere the meridian is the longitude).

Scale Factors (u,v)   These fields specify the scaling factors which will be applied to the u and v projections.

Show Cutting Seam   Click this button to display the current seam line location about the circumference of the revolved surface. The seam line is manipulated interactively via the Slider Bar.

Align with Parent Origin/Axis   Retrieves the Origin and Axis information from the Parent Part. Must be done if Parent Part is a quadric clip.

Feature Detail Editor
(Developed Surfaces

Double clicking on the Developed Surface Create/Update Icon opens the Feature Detail Editor for Developed Surfaces, the Creation Attributes Section of which provides access to all of the functions available in the Quick Interaction Area plus several more:

Figure 7-52
Feature Detail Editor (Developed Surfaces)

*Vector _|_ To Axis
Pointing To Seam*

These fields allow you to precisely specify the position of the Cutting Seam Line by specifying the direction of the vector perpendicular to the axis of revolution which points in the direction of the seam line.

Orig X Y Z

These fields specify a point on the axis of revolution.

Axis X Y Z

These fields specify a vector, which when used with the Axis Origin defines the axis of revolution.

The Feature Detail Editor also allows you to make changes in batch; that is, to make several changes to the menus and fields which do not effect the Graphics Window until you click in the Apply Changes button. It is sometimes quicker (with respect to CPU time) to make several changes at once rather than one at a time as in the Quick Interaction Area.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

(see How To Create Developed (Unrolled) Surfaces)

Troubleshooting Developed Surfaces

Problem	Probable Causes	Solutions
Error message is encountered while creating a Developed Surface Part.	Parent Part is invalid.	Only 2D Parts can be developed.
Developed Surface is created, but is either not visible, Partially visible, or obstructed by other Parts which may be other developed Parts	Since all Developed Surfaces are projected about the origin on the xy-plane of the reference frame of the parent Part, they may map outside the viewport, intersect other Parts, or pile up on each other.	Set the Developed Surface to be viewed in its own viewport and initialize the viewport. Use different u/v scaling. Assign the developed Part to its own local reference frame and transform it accordingly.
Developed Surface Part is a line.	Wrong Projection type was specified.	Select a different Projection.
Developed Surface Part does not update to new Orig and/or Axis values.	The Orig and Axis values can not be specified if the Parent Part is created from either a cylinder, sphere, cone, or revolution quadric clip. These values can only be specified if the 2D parent Part is not a quadric clipped surface.	Since values entered for this condition are not used, click the Get Parent Part Defaults button to update the fields based on the selected parent Part in the Parts & Frames list.

7.10 Displacements On Parts

Each node of a Part is displaced by a distance and direction corresponding to the value of a vector variable at the node. The new coordinate is equal to the old coordinate plus the vector times the specified Factor, or:

$C_{new} = C_{orig}$ + Factor * Vector,

where C_{new} is the new coordinate location, C_{orig} is the coordinate location as defined in the data files, Factor is a scale factor, and Vector is the displacement vector.

You can greatly exaggerate the displacement vector by specifying a large Factor value. Though you can use any vector variable for displacements, it certainly makes the most sense to use a variable calculated for this purpose. Note that the variable value represents the *displacement* from the original location, not the *coordinates* of the new location.

Clicking once on the Displacements On Parts Icon opens the Displacements Editor in the Quick Interaction Area which is used to specify how you wish to displace Part nodes based on a vector variable.
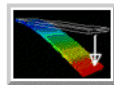

Figure 7-53
Displacements On Parts Icon

Figure 7-54
Quick Interaction Area - Displacements Editor

Displace by   This button allows selection of either None for no displacement or Variable (that selected in the Variables List) to use for displacement. The selected Variable must be a node-based vector and must be defined on the Parent Parts.

Displacement Factor   This field specifies a scale factor for the displacement vector. New coordinates are calculated as: $C_{new} = C_{orig}$ + Factor*Vector, where C_{new} is the new coordinate location, C_{orig} is the original coordinate location as defined in the data file, Factor is a scale factor, and Vector is the displacement vector. Note that a value of 1.0 will give you "true" displacements.

Apply New Variable   Changes the variable the Displacements are based upon to that currently selected in the Variables List.

Feature Detail Editor
(Model)

Double clicking on the Displacement on Parts Icon opens the Feature
Detail Editor for Model Parts, the Displacements Attributes turndown area of
which provides access to the same functions available in the Quick Interaction
Area.

(see Section 3.3, Part Editing for a detailed discussion of the other features
available in the Feature Detail Editor (Model)),

(see How To Display Displacements)

Troubleshooting Displacement Attributes

Problem	Probable Causes	Solutions
Displacement not visible	Displace By set to None for Part that is not displacing.	Set Displace By to Variable
	Displacement Factor value too small.	Specify a larger Displacement Factor.

7.11 Query/Plot

EnSight provides several ways to examine information about variable values. You can, of course, visualize variable values with fringes, contours, vector arrows, profiles, isosurfaces, etc.   This section describes how to query variables *quantitatively*:

Over Distance   EnSight can query variables at points over distance for the following information:
- variable values inside Parts at evenly spaced points along a straight line
- variable values inside Parts at the nodes of a different 1D Part

Over Time   EnSight can query variables over time for the following information:
- minimum and maximum variable values for Parts
- variable values at any number of sample times at any point inside of a Part or at any labeled node or element.

Over-time queries can report actual variable values, or Fast Fourier Transform (FFT) spectral values at the positive FFT frequencies.

Variable vs. Variable   EnSight can produce a scatter plot of one variable vs. another.

Operations on   EnSight can scale query values and/or combine one set of query values with another set to produce a new set of values.

I*mporting*   EnSight can import query values from external files.

Query Candidates   Only Parts with data residing on the Server host system may be queried. Thus, Parts that reside exclusively on the Client host system (i.e. contours, particle traces, profiles, vector arrows) may NOT be queried.

(see Section 3.1, Part Overview)

Clicking once on the Query/Plot Icon opens the Query/Plot Editor in the Quick Interaction Area which is used to query about the selected Variable on the selected Part and, if you wish, assign a query entity to a plotter.
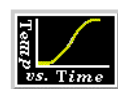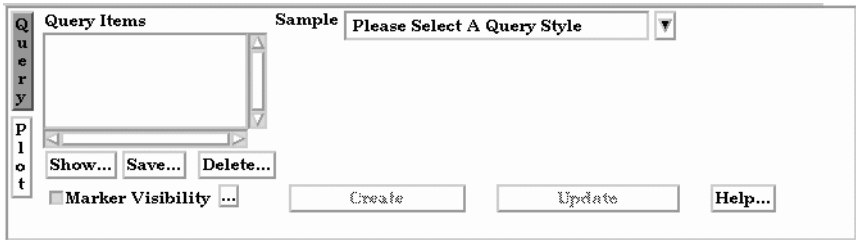

Figure 7-55
Query/Plot Icon

Figure 7-56
Quick Interaction Area - Query/Plot Editor

Query Items   This is the list of query items that currently exist in EnSight. After creating a query item, it will show up in this list and can be modified by selecting it in the list and changing the displayed values. (Note, it is best to deselect any query items in the list when creating a

new one, otherwise you may accidently modify values for the item with the left mouse button.)

Sample   This menu contains the types of queries that can be created. Selecting one of these changes the interface to display controls related to the type.

Figure 7-57
Query Sample Types
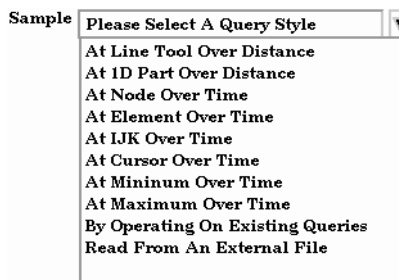
	Sample	Please Select A Query Style ▼

At Line Tool Over Distance
At 1D Part Over Distance
At Node Over Time
At Element Over Time
At IJK Over Time
At Cursor Over Time
At Mininum Over Time
At Maximum Over Time
By Operating On Existing Queries
Read From An External File

Please Select A Query Style   is displayed until a Sample selection is made.

At Line Tool Over Distance   queries at uniform points along the line tool.

At 1D Part Over Distance   queries at the nodes of a 1D part.

At Node Over Time   queries at a node over a range of times.

At Element Over Time   queries at an element over a range of times.

At IJK Over Time   queries at an IJK location over a range of times.

At Cursor Over Time   queries at the cursor tool location over a range of times.

At Minimum Over Time   queries the minimum of a variable over a range of times.

At Maximum Over Time   queries the maximum of a variable over a range of times.

By Operating On Existing Queries   forms new query by scaling and/or combining existing ones.

Read From An External File   imports previously saved or externally generated queries. (This can be EnSight XY data format or MSC Dytran .ths files.)

General to Each Type of Query

Marker Visibility   Toggles the visibility of the marker showing the location for the query. For distance queries, a sphere marker will be shown indicating the beginning location for the query.

...   Opens the Query Display Attributes dialog for the specification of the display attributes of the query marker.
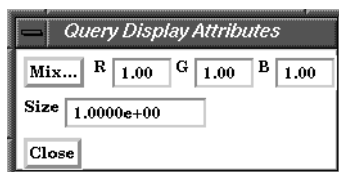

Figure 7-58
Query Display Attributes dialog

Mix...   Opens the Color Selector to specify the color of the marker.

RGB   The red, green, and blue color for the marker.

Size   The size of the sphere marker. The value is a scale factor. Values larger than 1.0 will scale the marker up, while values less than 1.0 (but greater than 0.0) will scale the marker down.

Update   This button causes the query to be recomputed using any modified attributes or variables.

Save...   Opens the Save Entity Query To dialog for the specification of the format, and file name in which you wish to save the query entity.
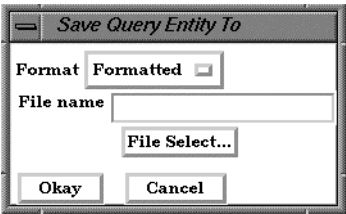

Figure 7-59
Save Query Entity To dialog

Format   Opens a pop-up menu to allow specification of the format. Choices are:

Formatted Outputs the query information to the specified file in the same format as the Show Text button.

XY Data   Outputs the query information in a generic format which could be used to export the information to a different plotting system.

File Name   This field is used to specify the file name in which you wish to save the query entity.

File Select   Opens up the File Selection dialog for specifying the File Name as an alternative to entering it manually in the File Name field.

Show ...   This button will display the results of the selected query in the EnSight Message Window.
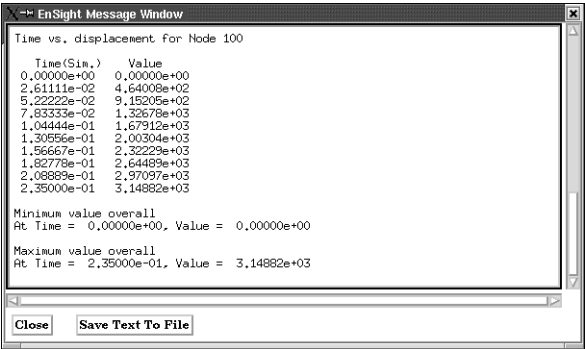

Figure 7-60
EnSight Message Window displaying query information

Save Text To File   Opens the File Selection dialog for specification of filename to save to.

Delete...   This button will delete the selected query items. You must confirm the deletion before it as actually done.

Create   This button will create the query according to options and variables specified.

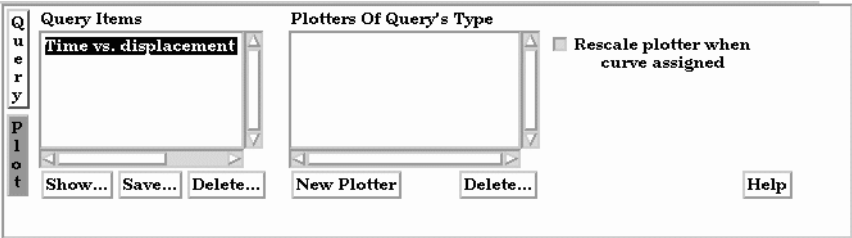**Plot**   Changes the Quick Interaction Area into the plotting section.

Figure 7-61
Plot Query Entity To dialog

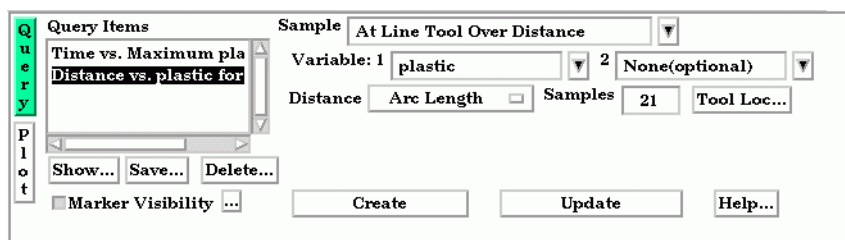|	
Query Items	Shows a list of current query items. As you select these, the plotters of the same type (if any) will be displayed in the Plotters Of Query's Type list. And if the query has been assigned to one of these plotters, it will be highlighted.
Plotters Of Query's Type	Shows a list of currently defined Plotters which are of the same type as the selected query. The selected query can be plotted on these or on a new plotter.
Rescale plotter when curve assigned	If a query is assigned to an existing plotter by selecting one in the Plotter Of Query's Type list, enabling this option will rescale the axis of the Plotter to include all queries that are assigned to it.
New Plotter	Will create a new plotter, add the new plotter to the list, and display the plotter with its query curve in the graphics window.
Delete ...	Will delete the selected plotters. You must confirm the deletion before it is actually done.

Query

At Line Tool Over Distance

Figure 7-62
Quick Interaction Area - Query/Plot Editor - **At Line Tool Over Diatance**

Variable: 1	A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis.
Variable: 2	If you leave this as "None", DISTANCE will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.
Distance	A menu of choices that control the distance parameter.

Arc Length	The distance along the part from the first node to each subsequent node (i.e. the sum of the 1D element lengths).
X Arc Length	The X coordinate value of each node accumulated from the start.
Y Arc Length	The Y coordinate value of each node accumulated from the start.
Z Arc Length	The Z coordinate value of each node accumulated from the start.
From Origin	The distance from the origin.
X from Origin	The X distance from the origin.
Y from Origin	The Y distance from the origin.
Z from Origin	The Z distance from the origin.

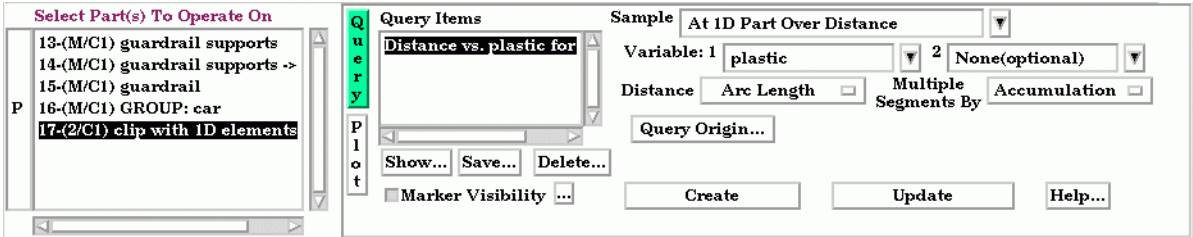|	
Samples	For queries over Distance using the Line Tool, this field specifies the number of equally spaced points to query along the line.
Tool Loc...	Brings up the Transformation Editor (Line Tool) dialog for feedback and manipulation of the location of the line tool.

At 1D Part Over Distance

Figure 7-63
Quick Interaction Area - Query/Plot Editor - **At 1D Part Over Distance**

Note that the 1D part to use for the query must be selected from the Part's list.

Variable: 1 A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis.

Variable: 2 If you leave this as "None", DISTANCE will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.

Distance A menu of choices that control the distance parameter.

Arc Length	The distance along the part from the first node to each subsequent node (i.e. the sum of the 1D element lengths).
X Arc Length	The X coordinate value of each node accumulated from the start.
Y Arc Length	The Y coordinate value of each node accumulated from the start.
Z Arc Length	The Z coordinate value of each node accumulated from the start.
From Origin	The distance from the origin.
X from Origin	The X distance from the origin.
Y from Origin	The Y distance from the origin.
Z from Origin	The Z distance from the origin.

Multiple Segments By When the selected 1D part contains more than one contiguous segment, these are handled by:

Accumulation	Each segment's query is appended to the previous. Thus a plot of this query will be one extended curve, but the extents of individual segement may not be obvious.
Reset Each	Each segment's query is treated like it is independent. Thus a plot of this query will appear as several curves.

Query Origin ... Brings up the Query Origin dialog for feedback and manipulation of the location of the query origin. .
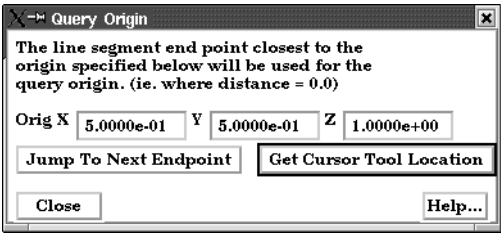

Figure 7-64
Query Origin dialog

Orig XYZ Coordinates of the location to use for query origin determination. The endpoint closest to the origin specified will be used as the "origin" of the query, i.e., where distance is zero. If the 1D part is s closed loop (i.e. there are no end points), the closest point on the loop is used as the "origin".

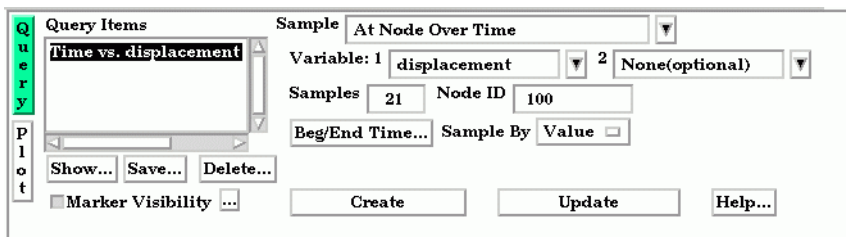| Jump To Next Endpoint | When multiple segments are present, clicking this button jumps to the beginning of the next segment, placing that location in to the Orig XYZ fields. |
| Get Cursor Tool Location | Places the current cursor tool location into the Orig XYZ fields so that point can be used as the query origin. |

At Node Over Time

Figure 7-65
Quick Interaction Area - Query/Plot Editor - **At Node Over Time**

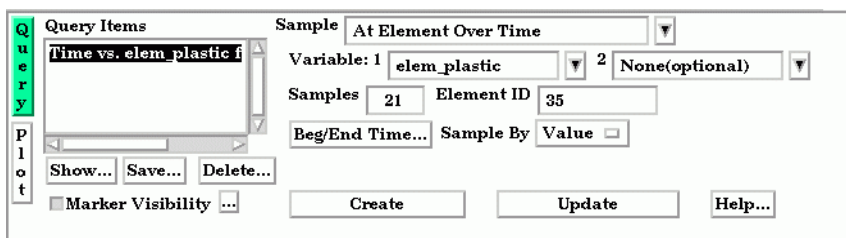| *Variable: 1* | A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis. |
Variable: 2	If you leave this as "None", TIME will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.
Samples	Specifies how many evenly timed moments over the specified range of time steps at which to query (if left blank, you get a sample point at each time step). If you specify more or fewer sample points than the number of time steps, EnSight linearly interpolates between the adjoining time steps. If query is an FFT sampling, the number of frequencies output will be less than or equal to the number of sample points.
Node ID	Specifies a node ID.
Beg/End Time ...	Opens up the Solution Time Editor in the Quick Interaction Area. Here you can specify the start and end times for queries Over Time. (see Section 7.13, Solution Time)
Sample By	Opens a pop-up menu for specification of how to report values for Over Time queries. Options are: *Value* reports values versus time. *FFT* reports FFT spectral values versus FFT positive frequencies.

At Element Over Time

Figure 7-66
Quick Interaction Area - Query/Plot Editor - **At Element Over Time**

Variable: 1	A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis. (Note: only per_element variables can be used for this query type.)
Variable: 2	If you leave this as "None", TIME will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.
Samples	Specifies how many evenly timed moments over the specified range of time steps at which

to query (if left blank, you get a sample point at each time step). If you specify more or fewer sample points than the number of time steps, EnSight linearly interpolates between the adjoining time steps. If query is an FFT sampling, the number of frequencies output will be less than or equal to the number of sample points.

Element ID   Specifies an element ID.

Beg/End Time ...   Opens up the Solution Time Editor in the Quick Interaction Area. Here you can specify the start and end times for queries Over Time.
(see Section 7.13, Solution Time)

Sample By   Opens a pop-up menu for specification of how to report values for Over Time queries. Options are:
Value reports values versus time.
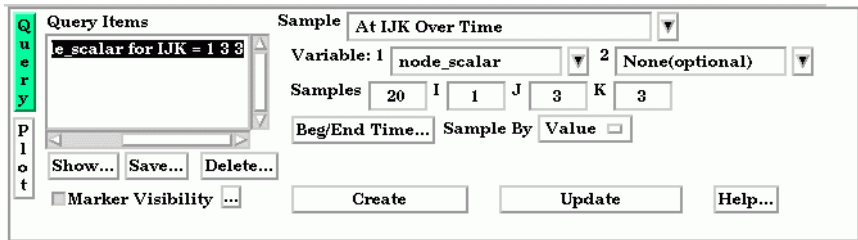*FFT*   reports FFT spectral values versus FFT positive frequencies.

At IJK Over Time

Figure 7-67
Quick Interaction Area - Query/Plot Editor - **At IJK Over Time**

Variable: 1   A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis.

Variable: 2   If you leave this as "None", TIME will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.

Samples   Specifies how many evenly timed moments over the specified range of time steps at which to query (if left blank, you get a sample point at each time step). If you specify more or fewer sample points than the number of time steps, EnSight linearly interpolates between the adjoining time steps. If query is an FFT sampling, the number of frequencies output will be less than or equal to the number of sample points.

IJK   Specifies the IJK planes of the desired location.

Beg/End Time ...   Opens up the Solution Time Editor in the Quick Interaction Area. Here you can specify the start and end times for queries Over Time.
(see Section 7.13, Solution Time)

Sample By   Opens a pop-up menu for specification of how to report values for Over Time queries. Options are:
Value reports values versus time.
FFT   reports FFT spectral values versus FFT positive frequencies.
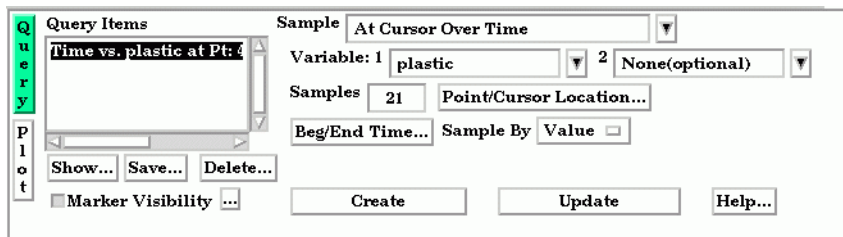
At Cursor Over Time

Figure 7-68
Quick Interaction Area - Query/Plot Editor - **At Cursor Over Time**

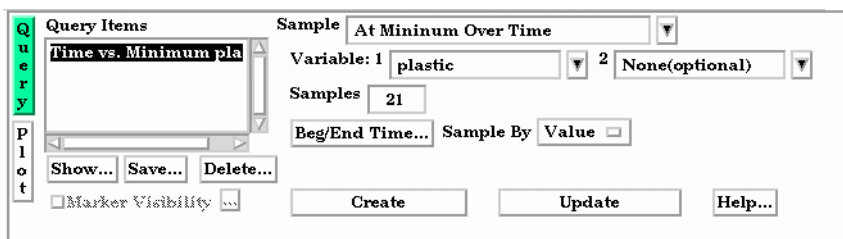|	
Variable: 1	A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis.
Variable: 2	If you leave this as "None", TIME will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.
Samples	Specifies how many evenly timed moments over the specified range of time steps at which to query (if left blank, you get a sample point at each time step). If you specify more or fewer sample points than the number of time steps, EnSight linearly interpolates between the adjoining time steps. If query is an FFT sampling, the number of frequencies output will be less than or equal to the number of sample points.
Point/Cursor Location ...	Can be used to open up the Transformation Editor (Cursor Tool) dialog for specification of the cursor location. You can of course also set this location using interactive or picking methods.
Beg/End Time ...	Opens up the Solution Time Editor in the Quick Interaction Area. Here you can specify the start and end times for queries Over Time. (see Section 7.13, Solution Time)
Sample By	Opens a pop-up menu for specification of how to report values for Over Time queries. Options are: *Value* reports values versus time. *FFT* reports FFT spectral values versus FFT positive frequencies.

At Minimum Over Time

Figure 7-69
Quick Interaction Area - Query/Plot Editor - **At Minimum Over Time**

Variable: 1	A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis.
Variable: 2	If you leave this as "None", TIME will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.
Samples	Specifies how many evenly timed moments over the specified range of time steps at which to query (if left blank, you get a sample point at each time step). If you specify more or fewer sample points than the number of time steps, EnSight linearly interpolates between

the adjoining time steps. If query is an FFT sampling, the number of frequencies output will be less than or equal to the number of sample points.

Beg/End Time ...   Opens up the Solution Time Editor in the Quick Interaction Area. Here you can specify the start and end times for queries Over Time.
(see Section 7.13, Solution Time)

Sample By   Opens a pop-up menu for specification of how to report values for Over Time queries. Options are:
Value reports values versus time.
FFT   reports FFT spectral values versus FFT positive frequencies.

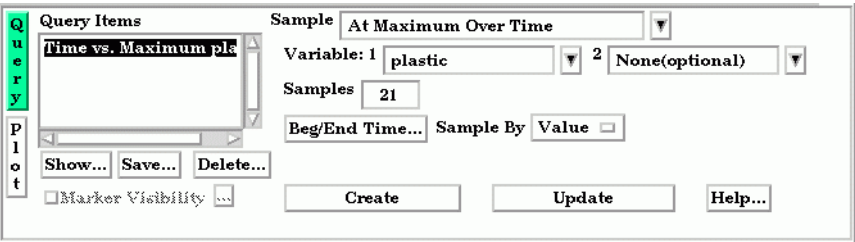### At Maximum Over Time

Figure 7-70
Quick Interaction Area - Query/Plot Editor - **At Maximum Over Time**

Variable: 1   A list of variables than can be chosen for the query. If plotted, this variable will be plotted along the Y-Axis.

Variable: 2   If you leave this as "None", TIME will be the default X- Axis variable. If you choose a variable form the list, a "scatter plot" query will result, and the X-Axis will be the variable you have chosen.

Samples   Specifies how many evenly timed moments over the specified range of time steps at which to query (if left blank, you get a sample point at each time step). If you specify more or fewer sample points than the number of time steps, EnSight linearly interpolates between the adjoining time steps.   If query is an FFT sampling, the number of frequencies output will be less than or equal to the number of sample points.

Beg/End Time ...   Opens up the Solution Time Editor in the Quick Interaction Area. Here you can specify the start and end times for queries Over Time.
(see Section 7.13, Solution Time)

Sample By   Opens a pop-up menu for specification of how to report values for Over Time queries. Options are:
Value reports values versus time.
FFT   reports FFT spectral values versus FFT positive frequencies.
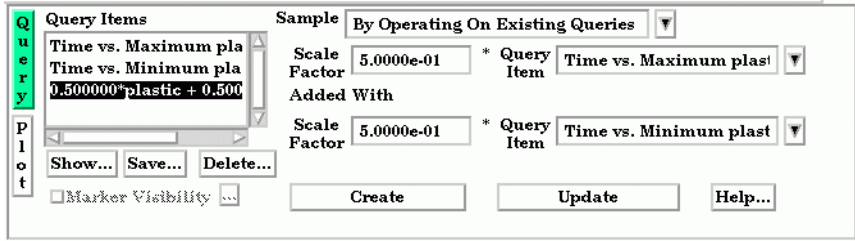
By Operating On Existing Queries

Figure 7-71
Quick Interaction Area - Query/Plot Editor - **By Operating On Existing Queries**

Scale Factor   Scale factor for the Query Item selected. The values of the selected query will be multiplied by this factor either before it is added to the second query or before the new query is created (if only operating on a single query).

Query Item   The existing query item(s) to operate on. A new query will be create consisting of scaled values one query, or the scaled, algebraic sum of two queries.
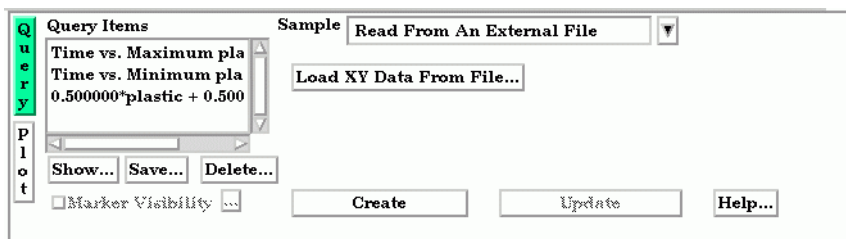
Read From An External File

Figure 7-72
Quick Interaction Area - Query/Plot Editor - **Read From An External File**

Load XY Data Opens the File Selection dialog from which a previously saved or externally generated
From File   query can be retrieved. EnSight's XY data format or MSC Dytran .ths files can be read.

(See also How To Query/Plot)

7.12 Interactive Probe Query

EnSight enables you to obtain scalar, vector, or coordinate information for the model at a point directly under the mouse pointer, at the location of the cursor tool, or at particular node, element, ijk, or xyz locations. The information is normally displayed in the Interactive Probe Query section of the Quick Interaction Area, but it can also be displayed in the Graphics Window. The performance of Interactive Query operations is dependent on the refresh time of the Graphics Window. Interactive query values are not echoed to EnSight command language files.

Clicking once on the Interactive Probe Query Icon opens the Interactive Probe Query Editor in the Quick Interaction Area which is used to specify parameters for querying interactively.
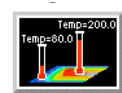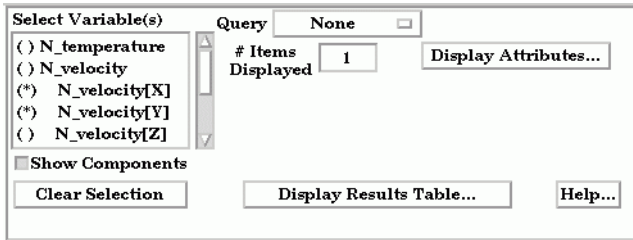

Figure 7-73
Interactive Probe Query Icon

Figure 7-74
Quick Interaction Area - Interactive Probe Query Editor

Select Variable(s)   List of variables and their components (if vector and Show Components is toggles on).

Query   Selection of whether interactive query is on, or which method to use to indicate input. *Surface Pick* will query the location under the mouse in the Main View. The query will be performed when the "p" keyboard key is pressed (when "Pick Use 'p' " is on) or whenever the mouse moves to a new location in the Main View (when "Continuous" is on).

Cursor will query the location indicated by the Cursor Tool in the Main View. The query will be performed when the "p" keyboard key is pressed (when "Pick Use 'p' " is on) or whenever the Cursor Tool moves to a new location in the Main View (when "Continuous" is on.

Node will query the node as specified in the "Node ID" field.

IJK will query the IJK node as specified in the "I J K" fields.

Element will query the element as specified in the "Element ID" field.

XYZ will query the x, y, z location as specified in the "x y z" fields.

None indicates that interactive query is off.

Search   Selects the location for the query. (Only active for Surface Pick and Cursor queries.)

Exact indicates that the query will occur at the location of the mouse.

Closest Node indicates that the query will "snap" to the node closest to the mouse.

Pick (Use 'p')   When the Action is Surface Pick or Cursor, controls whether the query will occur on a keyboard 'p' key press (when on) or will occur continuously - tracking the mouse location.

|	
Node ID	For Node Queries, specify the node id.
Element ID	For Element Queries, specify the element id.
# Items Displayed	Sets the number of query locations that are kept in memory and displayed to the user.
Clear Selection	Clears all the selected variables.
Display Results Table...	Opens the Interactive Probe Query Results Table dialog which shows a table of all selected variables as well as the current query type, the latest xyz coordinates, the latest ijk values (if applicable), and the latest node or element id (if applicable). Note that the contents of this dialog can be saved to a file by using the Save... button.
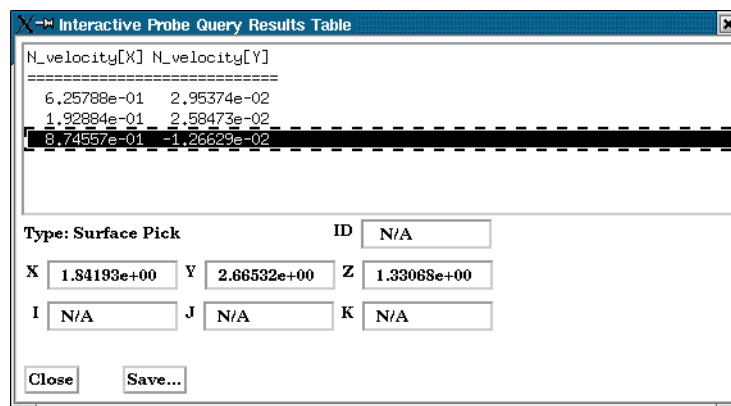
Figure 7-75
Interactive Probe Query Results Table

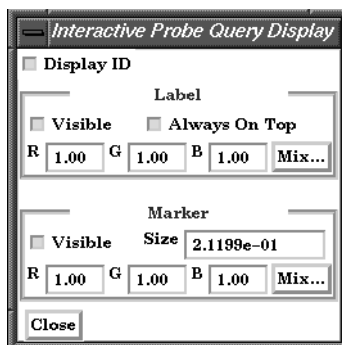*Display Attributes...*   Opens the Interactive Probe Query Display dialog.

Figure 7-76
Interactive Probe Query Display dialog

Display ID Toggle	When toggled on, if an ID is appropriate for the type of search, will display the ID in the query table and in the label on the model.
Label	
Visible Toggle	When toggled on, query information will be displayed in the Graphics Window
Always on Top	When on, query information in the Graphics Window will not be hidden from view behind other geometry.
RGB	These fields specify color values for the Labels.
Mix	Opens the Color Selector dialog
	(see Section 7.1, Color)
Marker	
Visible	When on, query location markers will be displayed in the Graphics Window.
Size	The size of the markers.
RGB	These fields specify color values for the markers.
Mix	Opens the Color Selector dialog
	(see Section 7.1, Color)

7.13 Solution Time

Many analyses contain time dependent information, such as automobile crash simulations and unsteady flow problems. The presence of time-dependent data is indicated to EnSight through an EnSight result file, case file, or is determined directly from the data files of other formats. EnSight has the capability of displaying the model and results at any time provided for in the data. Linear interpolation between given time steps is possible as long as the geometry does not have changing connectivity over time.

EnSight keeps track of which variables and Parts have been created so that if you change time steps, variables and Parts will update appropriately. For example, assume you have created a clip plane through the combustion chamber of an engine. From this clip plane you have created two constant variables Min Temperature and Max Temperature and are displaying them in the Main View. Now change time steps. First, the geometry updates to a new crank angle position. Second, the clip plane will automatically be recalculated to fit the new geometry. Third, the Min and Max values displayed in the Graphics Window are recalculated and updated. This is all performed automatically by EnSight after you change the current time value.

It is important to distinguish between time step and solution time. An example will best illustrate this concept.

Consider a model with data for 5 different times:

Time Step	Solution Time
0	1.0125
1	11.025
2	11.50
3	13.00
4	21.333

Note that the time steps coincide with the number of transient data files and are integers. The solution time at each time step comes from the analysis, and does not have to be at uniform intervals. The solution time can be in any units needed, but must be consistent with the solution files. That is, if a velocity file was in terms of meters per second, then the solution time must be in terms of seconds. Hence it is not possible, for example, to have the solution time reported in degrees crank angle for a combustion case unless the corresponding solution files were also in terms of crank angle (otherwise velocity would be reported in the meaningless units of meters-per-degree-crank-angle).

The Solution time must *always* be increasing in time. Failure to follow this rule will result in an error.

A special Solution Time dialog gives you control over time and relates time step to solution time. You can force the time information to conform to the actual time data given at the steps, or you can allow interpolation to occur between time steps. You must be aware of the implications of such an interpolation and choose the method that is appropriate.

Also, you can see the ranges of time dependent data available and the current time that is set for the Main View. You can change time steps by either entering a new time to view, or using the Solution Time slider bar.

The Solution Time Dialog shows a composite timeline of all timesets from all cases. For any case, a number of different timesets can exist. Each timeset can be attached to multiple variables and/or geometry. This makes it possible to, for example, have one variable defined at t = 1.0, 2.0, 3.5 and another variable defined at t = 1.5, 2.0, 4.0. For each timeline, controls exist to specify how EnSight should interpolate the variables when time is set to a value not defined for a given timeset.

There are other places within EnSight where time information is requested. These include, traces, emitters, animated traces, flip book transient data, key frame animation transient data, and Query/Plot. Each of these use the specified Beg/End values. For functions which do not explicitly specify the time step the current display time (as defined in the Solution Time Dialog) is used.

Clicking once on the Solution Time Icon opens the Solution Time Editor in the Quick Interaction Area which is used to specify time information.
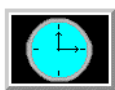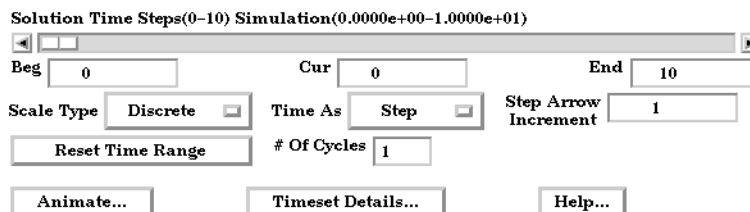

Figure 7-77
Solution Time Icon

Figure 7-78
Quick Interaction Area - Solution Time Editor

The range of both Time Steps and Simulation Time is shown at the top of the Editor.

Beg	Value for Beginning Time Step or Simulation Time depending on setting for Time As.
Cur	Value for Current Time Step or Simulation Time depending on setting for Time As. The slider bar can be used to select a value for the Current Time Step field.
End	Value for Ending Time Step or Simulation Time depending on setting for Time As.
Scale Type	Opens a pop-up menu to specify use of existing time steps, or allow EnSight to linearly interpolate to show any time step. Choices are: *Discrete* Can only change time to defined steps. *Continuous* Can change time to any time, including times between steps. Only available if do not have changing geometry connectivity transient case.
Time As	Opens pop-up menu to specify whether to use and display: *Steps* which will be an integer showing time as step data. Will show NOSTEP if in Continuous mode and current time is not at a given time step. Current time will automatically change to keep within range Begin/End range. The default beginning and ending simulation times correspond to the first and last time steps specified in the results.

Simulation Time which will be a real number showing true simulation time.
Current time will automatically change to keep within the Begin/End range. The default beginning and ending simulation times correspond to the first and last times specified in the results.

Reset Time Range   Will reset the Begin/End Time values to the minimum/maximum possible. Useful if you have specified your own Begin/End time values.

Step Arrow Increment Species the incremental time which will be applied to the current time each time the slider stepper buttons are used.

Animate Over Time...   Opens the Flipbook Animation Editor.
(see Section 7.14, Flipbook Animation)

# of Cycles   For cyclic transient analysis, the solution is often computed for one cycle only. It is often desirable to be able to visualize more than one cycle. This is possible only if the first and last timesteps contain the same information. By default, EnSight assumes one cycle.

Timeset Details...   will open the Timeset Details dialog.

Figure 7-79
Timeset Details Dialog

Which Timeset(s)   Selects the timesets to be viewed.

Modify All Selected Timeset(s)   Allows modification of all selected timesets.

Range which time range to modify.
Update Step Defn. To Choose how to modify the selected Timeset's Range.

Set Solution Time To Timeset Range   Will set the Solution Time Beg. and End. time values to correspond to the selected timeset.

Show Scale As   "Full Time Range" will show the Timeset's values in relation to the full composite timeline. "Timeset's Range" will adjust the beginning and ending boundaries of the graphic timeset to correspond to the begin and end values for the timeset. The change will not take effect until the "Update Selected Timeset(s)" button is pressed.

Defined For   Lists all of the variables and/or geometry attached to the Timeset.

Left/Right of   When the Current time is less than the Timeset's minimum time, the attached variables
Step Defn.   will use the Nearest values or become Undefined.

Between Steps   When the Current time is between the Timeset's minimum and maximum time values, but
Step Defn.   not defined, the attached variables will use the Right/ Left, Interpolate, or Nearest values,
or become Undefined.

Update Selected   Must be selected in order to update any changed Timeset.
Timeset(s)

7.14 Flipbook Animation

There are three common animation techniques which are easily accomplished with Flipbook Animation. They are:

- animation of transient data, which can be any combination of scalar/ vector variables, geometry, and discrete Particles

- animation of mode shapes based on a mode-shape displacement variable

- animation of a Part moving or changing value during animation, such as sweeping a 2D-Clip Plane or changing the value of an isosurface.

You can combine any of these techniques with the animation of Particle traces discussed in the previous Section 7.4.

The concept of a flipbook is similar to the stick figures you have probably seen in books where each page contains a picture. When you flip through the pages quickly you get the sense of motion. Flipbook animation stores a series of "pages" in Client memory which are then rapidly played back to create the illusion of motion. Pages can be loaded as *graphic images*, which may playback faster; or as *graphic objects*, which can be transformed after creating the flipbook, even while the flipbook is running.

For animation to be of interest, something must change from page to page. For *transient-data* flipbooks, you must have visualized something about the model that changes over time. For *mode-shape* flipbooks, you need to have set the displacement attributes of the Parts for which you want to see mode shapes (see Section 7.10 Displacement On Parts). For *created-data* flipbooks, you need to have used the Start/Stop utility or specified Animation delta values for the Parts.

The number of pages in the flipbook determines the length and smoothness of the animation. You directly or indirectly specify how many pages to create. While the Server performs the calculations, the Client stores the flipbook pages in memory. Just how many pages you can store depends on the amount of memory installed on your Client workstation. Your choice to load graphic images or graphic objects affects memory requirements, but the complexity of the model and the size of the Graphics Window determine which will use less memory in any particular situation.

You can control which original model Parts and created Parts will be updated for each time increment as the user chooses. This feature takes all dependencies into account. For example if an elevated surface was created from a 2D clip plane, the clip plane would be updated first and then the elevated surface based on the new clip. The ability to choose which Parts are or are not updated allows before and after type comparisons of a Part.

After creating the flipbook, options for displaying it include: running all or only a portion of it, adjusting the display speed, running under manual control or automatically, and running from the beginning or cycling back-and-forth between the two ends.

It is important to know that objects in the flipbook cannot be edited. If you wish to change something in the flipbook, you must reload it. If you decide to regenerate a flipbook (after changing something), you can choose to discard all the old pages, or keep any old page with the same page number as a new page.

This is very useful when you first load every tenth frame then decide to load them all. EnSight will not have to reload every tenth frame that already exists. When you are done with a flipbook, remember to click Delete All Pages. This will free up memory for other uses.

Flipbook vs. Keyframe   While you can implement any flipbook animation technique with keyframe animation (described in the next section), flipbook animation has three advantages. First, graphic-object-type flipbooks allow you to transform the model interactively to see from many viewpoints. Second, graphic-image-type flipbooks can be saved to a file and later replayed without having to have the dataset loaded, or even being connected to the Server. Third, the speed of display can be more interactive because the flipbook is in memory and can be flipped through automatically or stepped through manually.

Flipbook animation has a few disadvantages. First, you cannot change any Part attributes, except visibility and material properties, without regenerating the flipbook. Second, each page is stored in Client memory, which limits the number of pages and hence the duration of the animation.

Transient Data   Transient-data flipbooks have pages that correspond to particular solution times; i.e. step or simulation. You specify at which time value to start and stop the animation, and the time increment between each page. The time increment can be more than one solution-time value; this is useful in finding a range of interest or for a coarse review of the results. The increment can be a fraction, in which case the data for a page is interpolated from the two adjoining solution-time values.

Mode Shapes   Mode-shape flipbooks are used to show primary modes of vibration for a structure. This is done by using a per node displacement, enabling the Part to vibrate. While you can use any vector variable for a displacement, to see actual mode shapes you need to have a Results-file vector variable corresponding to each mode shape you wish to visualize. Note that you can create copies of Parts and simultaneously display them with different mode-shape variables, or one at its original state and the other with displacement for comparison.

The first page of a mode-shape flipbook shows the full displacement (as it is normally shown in the Graphics Window). The last page shows the full displacement in the opposite direction. The in-between pages show intermediate displacements in proportion to the cosine of the elapsed-time of the animation.

Created Data   Created-data flipbooks animate the motion of 2D-Clips and Isosurfaces according to their animation attributes. This animation allows you to show clipping planes sweeping through a model or to show a range of Isosurface values. The first page shows the Part's location as it appears in the normal Graphics Window. On each subsequent page, each 2D-Clip is regenerated at the new location found by adding the animation-delta displacement to the 2D-Clip's location on the previous page. Also, each Isosurface is regenerated with a new iso-value found by adding the animation-delta increment to the iso-value of the previous page.

Linear Load   Linear-loaded flipbooks are used to animate a displacement field of a part by linearly interpolating the displacement field from its zero to its maximum value. The variable by which the part is colored also updates according to the linearly displaced values. Like Mode Shapes, this utilizes a per node displacement. The function can be applied to any static vector variable.

Clicking once on the Flipbook Animation Icon opens the Flipbook Animation Editor in the Quick Interaction Area.
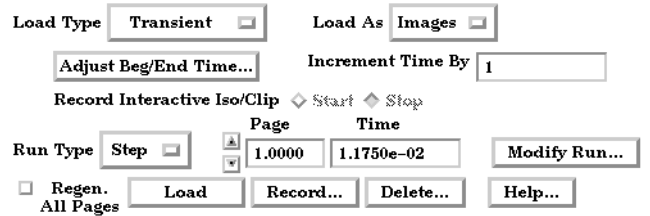

Figure 7-80
Flipbook Animation Icon

Figure 7-81
Quick Interaction Area - Flipbook Animation Editor

Load Type
Opens a pop-up menu for the selection of type of flipbook animation to load. Options are:

Transient   animates changes in data information resulting from changes in the transient data. For example, changes in coloration resulting from changes in variable values, or changes in displacement of Parts. See discussion in the introduction section.

Mode Shapes   animates the mode shape resulting from a displacement variable. See discussion in the introduction of this section.

Create Data   animates Parts having nonzero animation-delta values or which have been recorded with the Start/Stop utility. See discussion in the introduction of this section.

Linear Load   animates the Displacement (vector) variable of a part by linearly interpolating the displacement field from its zero to its maximum value. The Color variables of the part also update according to the linearly displaced values.

Load As
Opens a pop-up menu for the selection of whether to load flipbook pages as Graphic Images or Graphic Objects.

Graphic Objects flipbooks enable you to transform objects after creating the flipbook. Playback performance depends on the complexity of the model.

Graphic Images   flipbooks may be saved for later recall, but they cannot be transformed, nor can the window be resized. Playback performance depends on the Graphics Window size.

Adjust Beg/End Time...
Opens the Solution Time Editor in the Quick Interaction Area. To return to the Flipbook Animation Editor from the Solution Time Editor, click on Animate Over Time... (When loading Transient data, the flipbook will start and stop at the Beg/End values specified in the Solution Time Editor.
(see Section 7.13, Solution Time)

Increment Time By
In this field you specify the increment of each transient-data flipbook page which corresponds to the range type specified in the Solution Time Editor,

Note:
If you enter a Begin, End, or Increment value not corresponding exactly to a Step or Simulation time value, EnSight will interpolate the values, affecting the appearance of each page.

Record Interactive Iso/Clip
Allows you to define the change (isovalue change or clip plane movement) in an isosurface or clip plane which will take place during the Flipbook load. Only isosurfaces and clip planes which are modified in interactive mode are tracked.

Start - Stop	Start and stop the recording of interactive movement of isosurfaces or clip planes. Any interactive isosurface or clip plane modified between the Start and Stop will be modified during the flipbook load.
Run Type	Opens a pop-up menu for selection of how the loaded flipbook will play *Auto* makes flipbook play continuously. *Step* activates Page and Time fields and stepper buttons for manual page control. *Off* deactivates animation.

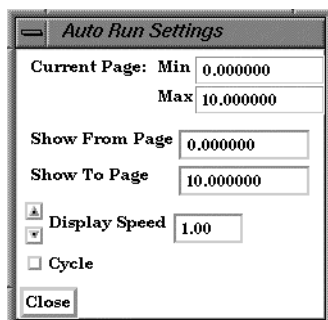Modify Run...   Opens the Auto Run Settings dialog.

Figure 7-82
Auto Run Settings dialog

You use the Flipbook Run Settings dialog to change the number of pages displayed, the running speed, and whether or not the flipbook playback repeats from the beginning or cycles playing forward and backward.

Current Page Min/Max	These fields display the minimum and maximum flipbook-page numbers currently loaded.
Show From Page Show To Page	These interactive fields specify the starting and ending flipbook pages to show when running flipbook.
Display Speed	This field specifies the playback-speed factor. Varies from 1.0 (full speed of your hardware) to 0.0 (stopped). Change by entering a value or clicking the stepper buttons.
Cycle Toggle	Toggles-on/off whether, during automatic playback, to replay from the beginning (toggled-off) or alternate playing forward and backward (toggled-on).
Regen. All Pages	Toggles-on/off whether to regenerate already created flipbook pages. When toggled-on, all
Toggle	existing pages are overwritten. When toggled-off, existing pages are not replaced by new pages having the same time value, and, if loading transient data, new pages can be interleaved according to their solution-time value.
Load	Clicking this button starts the loading flipbook pages and opens a pop-up dialog which reports the progress of the load and then closes to signal load is complete. If you cancel the load, the pages already created during the load remain in memory.

Record...   Opens the Save Flipbook Pages To dialog where you specify the type and name of the
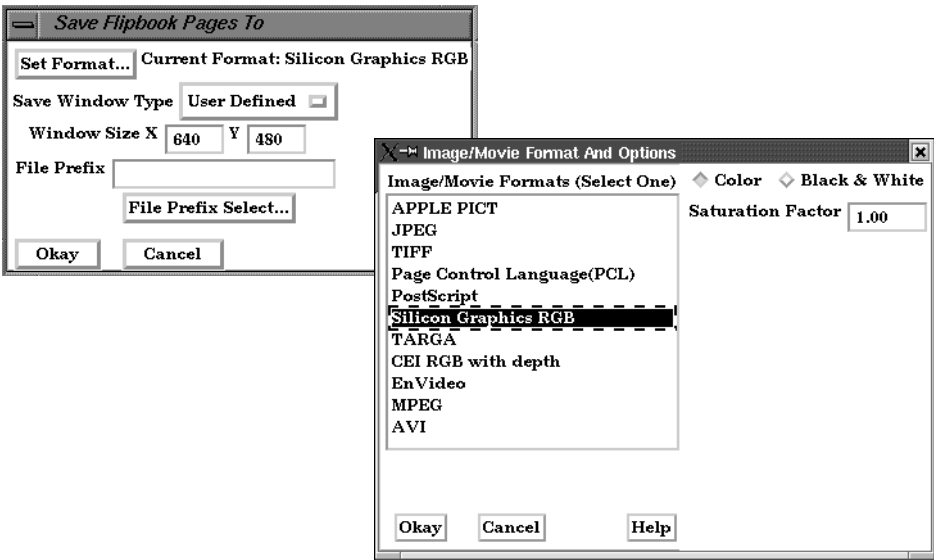   file in which you wish to save the flipbook animation pages you have created.

Figure 7-83
Save Flipbook Pages To dialog

Set Format...   Brings up the Image/Movie Format and Options dialog. The flipbook will be recorded
                                   using the selected format. (see Section 2.10, Saving and Printing Graphic Images and How
                                   to Print/Save an Image)

Save Window Type   When "Normal", will save images of the same size as the Main View. When "User
                                   Defined", will allow a specified width and height.

File Prefix   The location and filename prefix for the recorded images. The appropriate suffix will be
                                   added automatically.

Delete...   Opens a pop-up warning dialog which asks you if you really wish to delete all
                                   loaded pages. Click Okay to delete all loaded flipbook pages and free the memory
                                   for other use.

Troubleshooting Flipbook Animation

Problem	Probable Causes	Solutions
No motion	No pages are loaded.	Load flipbook pages.
	All pages are the same visually.	In order to see motion there must be a difference between one page and the next. Reload with differing Part attributes, such as coloring by a variable, using displacements, etc.
	Run Type set to Step or Off	Select Run Type to be Auto
Speed too fast	Display Speed is set too fast.	Change speed.
Speed too slow	Display Speed is set too slow.	Change speed.
	Hardware bottleneck (computer simply isn't sufficiently powerful)	Reduce the number of pages.Load pages as graphic images.
Speed erratic	Virtual memory is swapping pages to and from disk storage.	Only load the no.of pages that fit into the workstation's main memory.
Mode Shape(s) not visible	Wrong Load Type setting	Change Load Type to Mode Shapes and reload.
	Displacement attributes are incorrect.	Change Displace by and Factor attributes for the Part to animate.
2D Clip plane(s) not moving	Wrong Load Type setting	Change Load Type to Mode Shapes and reload.
	Plane was not moved interactively between Start and Stop.	
Isosurface(s) not moving	Wrong Load Type setting	Change Load Type to Mode Shapes and reload.
	Isosurface was not moved interactively between Start and Stop.	
Transient data ignored	Wrong Load type	
	Solution time step specifications are incorrect.	Change Load Type to Transient and set Solution time values according to available time steps.
Pages lost	Show From or Show To pages are not at ends of flipbook.	
	Old pages are being regenerated.	Toggle-off Regen. All Pages
	Delete All Pages is clicked.	Recover using the session command file.
Transformations do not work	Flipbook pages are loaded as graphic images.	Reload flipbook pages as Graphic Objects

7.15 Keyframe Animation

Since its initial release in 1987, EnSight has been used extensively for animation, due to its easy-to-use keyframe animator, ability to handle transient data, and ability to communicate with common animation controllers. This mechanism allows you to create your own movie sequence to present your results more easily. There seems to be two mind sets when it comes to animation. The first group of people believe animation to be totally trivial—something that can be completely finished in an hour or two by anyone. The other group of people seem to believe that animation is something that takes many days, if not weeks to finish and requires an "animation expert" to get done. Well, neither of these ideas are correct. While animation is not trivial, it is also not overly complicated. Most animation produced by EnSight is setup during a day, and recorded the same day or during the night to be complete by the next morning. Engineers create and record their own animations. The majority of the time involved takes place in the recording of the frames to the recording device. EnSight is intended to be used by end users— this includes the animation module. We do acknowledge, however, that there is a difference between animation, and animation done well. The latter comes with time and experience.

EnSight uses a modified keyframe technique. This technique enables the user to define what the scene should look like at certain times called Keyframe. Each keyframe can be different from a previous keyframe by using any combination of rotate, translate, scale, zoom, look-at, or look-from operations. A given keyframe can also be the same as the previous frame (the purpose of which will be explained shortly). The keyframe technique only works on transformations, and is not used for other items related to what the scene looks like (i.e., when to turn on Parts, do isosurfaces, shading changes, etc.). EnSight actually keeps track of the transformation commands performed between keyframes and linearly interpolates these commands when creating frames between the keyframes. These in-between frames are referred to as subframes. A reset command will not be allowed during keyframe animation because it cannot be interpolated.

Each keyframe includes the following information: (1) a set of transformation matrix values, specifying each local frame, the global frame, the Look-At and Look-From Points, and the position of the Plane Tool; (2) the value of all isosurfaces and position of all clip Parts using the plane tool; (3) the specific keyframe attributes; and (4) the transformation commands and isosurface values to get the scene and clip Parts to the next keyframe.

When running keyframe animation, EnSight performs the following actions for each keyframe: (1) any command language commands associated with the keyframe are executed, (2) the specified number of subframes are displayed in sequence, interpolating among them the *resultant* of the transformations *you performed* to get to the next keyframe, which are *not* necessarily the most direct transformations to get to the next keyframe.

What is meant by the *resultant* of your transformations? Consider, for example, if you create a keyframe, rotate the model 360 and create another keyframe, when you run the animation the model will rotate 360, not stay still. But if you create a keyframe, rotate the model one way 360, rotate it back the other way 360 and then create an keyframe; when you run the animation the model will not move because the *resultant of your transformations* produced no rotation.   If you had created a

keyframe between the two transformations, then both rotations would have been seen. The various types of transformations are performed simultaneously.

To begin the process of creating an animation sequence, first define the scene you desire for the first keyframe. This includes having all the Parts you want shown, having the attributes you wish for these Parts, orienting as desired, etc. Then, turn on keyframe animation and create this scene as your first keyframe. You can then proceed to modify the orientation of the model and create your other keyframes.

If you make mistakes during the keyframe definition, click Delete Keyframe ... and enter the number of the last keyframe you were satisfied with. Then, proceed to define the subsequent keyframes again. As soon as you have at least two keyframes defined, you may play back the animation to see what it looks like. To do this, select the Run Animation button in the Quick Interaction Area. The animation process generally proceeds with some keyframe definitions, running what you have so far after some of those definitions, once in a while a delete back to operation, more keyframe definitions, etc., until you are satisfied with the entire animation sequence. You then set up the recording device information and set the process in motion to produce a video.

Note, that when playing back the animation, you do not have to always play the entire sequence. Run From, and To frame capability is provided. You also can abort an animation run by entering the "a" key in the graphics window.

In order to get the length of animation you want on video, you will need to adjust the number of sub-frames between keyframes in the Keyframe Speed/Actions dialog. The total number of frames displayed during animation is the sum of the keyframes plus the sum of the subframes. The NTSC broadcasting standard calls for 30 frames displayed per second. On most workstations, it is unlikely that EnSight will be able to display this rapidly during playback on the workstation. So it can be difficult to get a feel for how fast the animation will be once recorded. The speed of the playback on the workstation is related both to its graphics capability and the complexity of the scene, so reducing the complexity will speed things up. Accordingly, you might consider options like making all but a representative Part invisible, use the feature angle option to reduce the visual complexity of the Parts, and/or use the dynamic/static box drawing modes.

Anything that is currently on will be on during the animation. That is, if contours, vector arrows, Particle traces, shaded surfaces, flipbook animation, animated traces, etc. are on, they will be on during animation. If any Parts have an animation delta set or are dependent on a Part that has the delta set then they will be regenerated and change through the animation. This enables you to do any of the flipbook animation techniques within keyframe animation for recording purposes, including the use of transient data (See Flipbook Animation). The advantage for doing flipbook techniques within keyframe animation is that they can be recorded and the amount of memory used is smaller because the whole flipbook is not loaded into memory. This enables the recording of long sequences of changing information that would not be able to be shown fully with flipbook animation because of memory limits of the workstation. Short sequences that you have already loaded into the flipbook can also be used by making sure that the Flipbook Run toggle is on before running keyframe animation.

If dealing with transient data, you should set up the keyframes for display of the model first, play it back, edit, etc. Then, after you are satisfied with the model presentation, you can start dealing with displaying the transient data on the model. You should be careful in doing movement of the model while transient data is being displayed. It can be confusing to have the transient data changing at the same time that the model in the scene is moving. When dealing with transient data, we normally introduce the problem first with some keyframes, then run the transient data without any transformations by defining two successive identical keyframes. Between these two identical keyframes, we animate the transient data using one of the several methods available.

We have attempted to create the animation module to be able to run in a set-up-walk-away mode to create video. In order to do this, you can issue command language lines at each keyframe. For example, if you had a case where you wanted to first show off some of the model, and then turn on fringes to show results, you could issue a "view: fringes on" command at the keyframe. It is also possible to play a command language file using this option. Care should be taken to *not* issue an `anim_keyframe: run` command as Part of this command language (which would cause an infinite loop).

When saving images to disk files, be aware that image files can take a great deal of disk space. The file system that you are writing images to should be monitored during the animation run to make sure it doesn't run out of space.

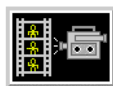Clicking once on the Keyframe Animation Icon opens the Keyframe Animation Editor in the Quick Interaction Area.
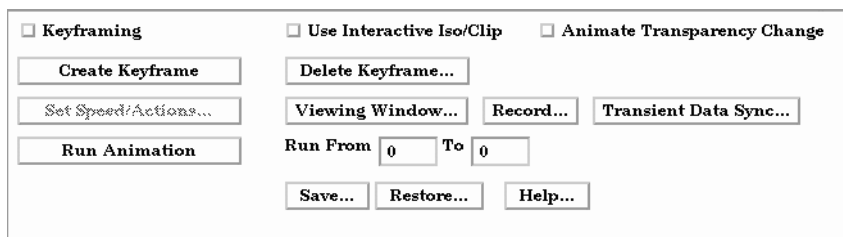

Figure 7-84
Keyframe Animation Icon

Figure 7-85
Quick Interaction Area - Keyframe Animation Editor

Keyframing Toggle   Toggles-on/off Keyframe animation feature. WARNING: If you toggle-off Keyframing, all the keyframes previously created will be lost (see Save... below).

Use Interactive Iso/Clip   By turning this toggle on, any clip or isosurface moved during the keyframe will animate.

Animate Transparency Change   By turning this toggle on, transparency changes to parts during the definition of the keyframes will be part of the animation.

Create Keyframe   Click this button to create a keyframe. If Keyframing toggle is not turned on then creating the first keyframe will turn it on automatically. Keyframes are automatically numbered in sequence of their creation. As each keyframe is created, a message appears in the Status History Area.

Delete Keyframe…	Opens the Delete Keyframes pop-up dialog which allows you to specify the number of the last keyframe you wish to retain and then delete all keyframes back to that frame. The keyframe whose number you specify is not deleted. To delete all keyframes enter 0 at the prompt.
Run Animation	Click to run the keyframe animation. If you click Run more than once, the animation will play for the corresponding number of times. To abort the run, press the "a" key in the Graphics Window.
Run From *Run To*	These fields specify the numbers of the keyframe to start from and the keyframe to run to when Run button is pressed. Must be integer numbers of already created keyframes. Default is Run From 1 and Run To number of keyframes you have created.
Set Speed/Actions...	Opens the Keyframe Speed(Subframes)/Actions(Commands) dialog.
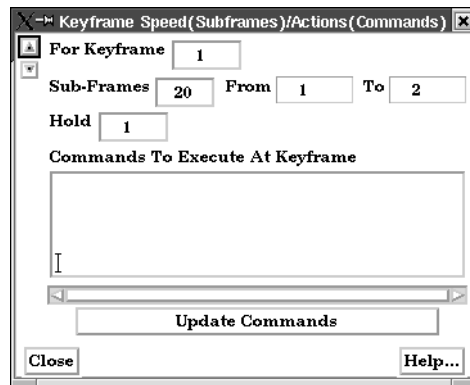
Figure 7-86
Keyframe Speed(Subframes)/Actions(Commands) dialog

For Keyframe	This field and the stepper buttons are used to select which keyframe to edit.
Sub-Frames From To	The Sub-Frames field specifies the number of subframes between that keyframe specified in the "From" field and that specified in the "To" field. More subframes make the transformations to the next keyframe smoother and slower.
Hold	This field specifies the number of frames to hold at the keyframe.
Commands to Execute at Keyframe	This command text area is used to specify up to five commands to execute before displaying the keyframe referenced in the For Keyframe field. You may use any command except commands corresponding to nonpermitted actions, such as loading another dataset. Also, there is no point in using `view_transf` commands that transform frames, change the Look At and Look From points, or move the Plane Tool since the next thing EnSight does is update the Graphics Window to match the transformation matrix information stored as Part of the keyframe. You may use `anim_keyframe` commands, for example, to toggle-on using transient data, but you should not use the `anim_keyframe: run` command since then the animation will enter an infinite loop. Commands frequently used here would be `view:` and `annotation:` commands. You may also play a command file, so there is really no limit as to how many commands you can execute. The `shell:` command is a special command to issue a UNIX command.
Update Commands	This button will accept the commands entered above.

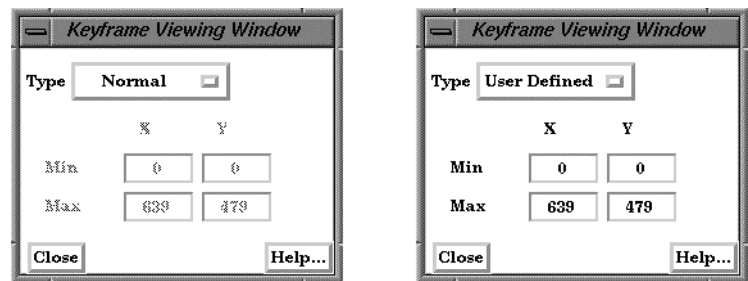*Viewing Window...* Opens the Keyframe Viewing Window dialog.

Figure 7-87
Keyframe Viewing Window dialog

Type Selection of image type, including standard video formats. Options are:
Normal type is appropriate for display in the Graphics Window.
Full type is appropriate for a full-screen graphics window.
NTSC type is NTSC window size.
PAL type is PAL window size.
User Defined type enables you to specify the screen dimensions (see below).

Min X Y In Type: User Defined, these fields allow you to specify the pixel dimensions for user-
Max X Y defined type of screen. Allowed values for X are 0 to 1279 and for Y from 0 to 1023.
Bottom left corner is 0,0. EnSight assumes a horizontal-to-vertical aspect ratio of 5-to-4.
Other aspect ratios will distort the images.

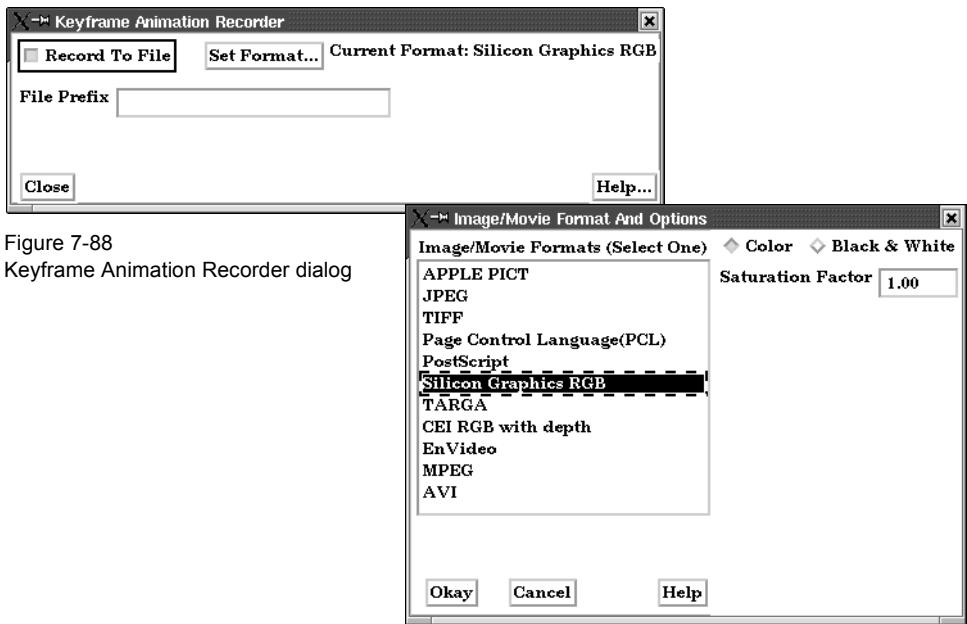*Record...* Opens the Keyframe Animation Recorder dialog.

Figure 7-88
Keyframe Animation Recorder dialog

Record To File When on, will record the keyframe animation images to the specified filename(s).

Set Format... Brings up the Image/Movie Format and Options dialog. The flipbook will be recorded
using the selected format. (see Section 2.10, Saving and Printing Graphic Images and How
to Print/Save an Image)

Render to On some SGI models it is possible to render the keyframe animation in the "background",
Offscreen Buffer thus freeing the workstation for other tasks.

For Keyframe This field and the stepper buttons are used to select the number of frames to Hold. Each
Hold will record the indicated number of frames to the Record To device.

File Prefix	Specification of filename prefix to use when saving frames to a disk file. Each frame is saved in a different file according to the File Numbering. The prefix can also have a directory path before it, such as /usr/tmp/prefix.
File Numbering	Opens a pop-up menu for the selection of file numbering method. Choices are:

Frame Code File names will contain the keyframe and sub-frame numbers as indicated under File Prefix.

Sequential File names will be numbered from 1 sequentially upwards throughout the animation.

Transient Data Sync... Opens the Keyframe Transient Data Synchronization dialog.
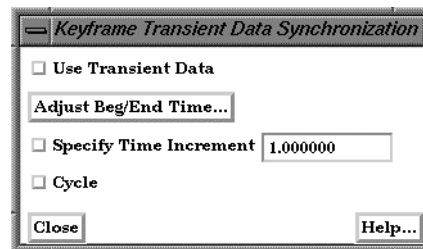
Figure 7-89
Keyframe Transient Data Synchronization dialog

Use Transient Data Toggle	Toggles-on/off synchronization of keyframe animation to transient data. If toggled-on, each animation frame (keyframe or subframe) will cause the next transient-data time to be swapped in. You can interpolate between whole time steps with the Specify Increment attribute (see below).
Adjust Beg/ End Time...	Opens the Solution Time dialog to allow modification of the ranges. (see Section 7.13, Solution Time)
Specify Time Increment	Toggle-on/off to indicate that you want to specify how to step through the transient data, or that you want EnSight to calculate how to do this. For example, for a Step time, entering
Toggle / Field	1 indicates that you want EnSight to step by 1 with no interpolation. To cause frames to be interpolated between actual time steps then enter a fractional value, such as 0.5 to interpolate half way between or 0.25 to interpolate at each 1/4. Enter a value greater than 1 to skip over transient times, such as using 10 to jump by tens.
Cycle Toggle	Toggles-on/off cycling through transient data during playback of keyframe animation. Toggling-on cycling will cause the transient data to be played from beginning to end and then end to beginning depending on how many keyframes and subframes you have specified.
Save…	Opens the File Selection dialog to allow you to save the specifications of the current keyframe into a file. This saves only the keyframe specifications, not the animation images or Part information. If you perform a Full Backup, the keyframe specifications are saved as Part of the Backup.
Restore…	Opens the File Selection dialog to allow you to restore keyframe specifications from a file. This restores only the keyframe specifications; you must also load Part data and set the Part attributes.
Use Interactive Iso/Clip	Turn on to animate clip and isosurface parts which were interactively modified during the keyframe animation.

(see How To Create a Keyframe Animation)

Troubleshooting Keyframe Animation

Problem	Probable Causes	Solutions
Graphics Window flashes at start of animation run.	New graphics window is opened to display the animation.	Hardware specific. Does not affect frames sent to recorder.
Colors seem to bleed when I play the recorded tape back.	The color display from tape has a tendency to bleed colors that are pure, such as full intensity red, green, or blue.	Don't use them. If you want a red object don't use full intensity, and mix in some other components. For example you may want to try RGB =.9 .1 .1.
Lines "crawl" across the screen when I play the recorded tape back.	Lines are only 1 pixel wide which would cause crawling on video tape.	Use a line width greater than 1.
During playback the action of the video starts as soon as the picture comes up and it's hard to recognize what is happening that quickly and then it goes away.	When creating a video it is best to have the model come up with a hold of 3 seconds or more before starting the animation. The animation should run for a reasonable length of time and then it should hold for 3 or more seconds again at the end. On complex models the hold may need to be as much as 10.	Holding a video at the beginning and the end and showing enough frames in-between will allow your audiences eyes to adjust and increase comprehension of the video. Adding annotation strings and pointers to point out areas of interest also helps. Also, showing the whole model with a hold and then zooming way in on the area of interest will help comprehension.
Video is too fast when played back from a recorded tape, but it looked fine on the monitor.	Video formats play back at rates that are normally faster than the workstation hardware can. For example, NTSC plays back at 30 frames per second which can be impossible for the workstation to match on a fairly complex model.	Increase the number of frames recorded by adding more subframes or by possibly having your video recording device record more than one frame when EnSight tells it to record.
Transformation of my object on the recorded tape is not smooth.	Not enough subframes.	Adding more subframes will cause more finely interpolated scene between keyframes. For instance the model should probably not rotate more than 3 degrees between frames being recorded.
Model is being clipped away as the animation proceeds.	Running into the Z-Clip plane or the regular plane tool with Clipping on.	Make sure the Z-Clip planes and the plane tool are far enough away from the model for the whole animation sequence. NOTE: The distance between the Z-Clip planes could affect the clarity of the image. The Z-Clip should be kept as close to the model and as close to each other as possible for better results.

7.16 Subset Parts Create/Update

EnSight enables you to create and modify Subset Parts from ranges of node and/or element labels of model parts. The Subset Parts feature allows you to isolate contiguous and/or non-contiguous regions of large data sets, and apply the full-range of feature applications and inspection provided by EnSight.

Subset Parts can only be created from parts that have node and/or element labels. Therefore, Subset Parts can not be created from any Created Parts, because the only parts that can have node and element labels are Model Parts such as parts built from file data, Merged Model Parts, or Computational Mesh Model Parts (parts created via the periodic computational symmetry Frame attribute). Model Parts that do not have given or assigned node and/or element labels can not be used to create Subset Parts.

Subset Parts are created and reside on the server. They are Created Parts that provide proper updating of all dependent parts and variables.

Subset Parts are created and modified by specifying parent parts, as well as their node and/or element labels. Node and/or element labels can be displayed and filtered interactively according to global View Mode and local Part Mode attributes.

Clicking once on the Subset Parts Create/Update Icon open the Subset Parts Editor in the Quick Interaction Area which is used to both create and update Subset Parts.
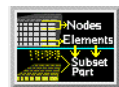

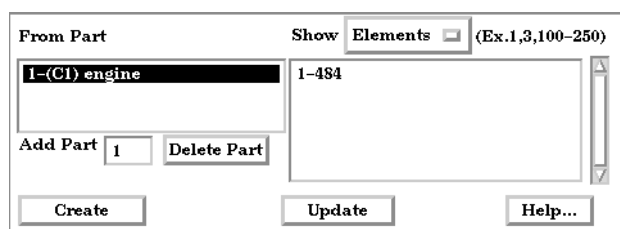Figure 7-90
Subset Parts Create/Update Icon

Figure 7-91
Quick Interaction Area - Subset Parts Editor

From Part   List reflecting the parent parts that have been added to the list. Selecting a part in this list displays any corresponding element or node range specifications in the Show List.

Show   Opens a pull-down menu for selecting which type of part entity you wish to include (or have included) in your Subset Part. The Show options are:
Elements show any specified element label ranges
Nodes   show any specified node label ranges

Show List   This field specifies the label ranges of Elements and/or Nodes wanted for the Subset Part that correspond to the selected part in the From Part list. The Elements or Nodes are specified as a range as the example indicates, i.e. (Ex. 1,3,8,9,100-250).

Add   This field specifies the GUI part number you wish to add to the From Part list.

Delete   This button deletes any selected entries in the From Part list along with any corresponding element or node range specifications in the Show List.

Update Part   Recreates the Subset Parts selected in the Main Parts List according to the selections in the From Part List and the Show List.

Feature Detail Editor (Subset Parts)   Double Clicking on the Subset Parts Create/Update Icon brings up the Feature Detail Editor (Subset Parts), the Creation Attributes of which offers the same features for the Subset Parts as the Quick Interaction Area Editor.

(see Section 3.3, Part Editing for a detailed discussion of the remaining Feature Detail Editor turn-down sections which are the same for all Parts),

7.17 Tensor Glyph Parts Create/Update

Tensor glyphs visualize the direction and tension/compression of the eigenvectors at discrete points (at nodes or at element centers) for a given tensor variable.

Tensor glyph Parts are dependent Parts known only to the client. They cannot be used as a parent Part for other Part types and cannot be used in queries. As dependent Parts, they are updated anytime the parent Part and/or the creation tensor variable changes (unless the general attribute Active flag is off).

Tensor glyphs can be filtered to show just the tensile or compressive eigenvectors. Further, the visibility for each of the eigenvectors (Major, Middle, and Minor) can be controlled.

Tensor glyphs will appear for each of the nodes/elements for the Parent part's visual Representation. Thus, for a border Representation of a Part, only the border nodes/elements will be candidates for a tensor glyph.

The tensile and compressive eigenvectors can be visualized by modifying the tensile/compressive component's line width and color.

Clicking once on the Tensor Glyph Create/Update Icon opens the Tensor Glyph Editor section of the Quick Interaction Area which is used to both create and update (make changes to) tensor glyph Parts.
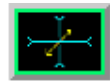

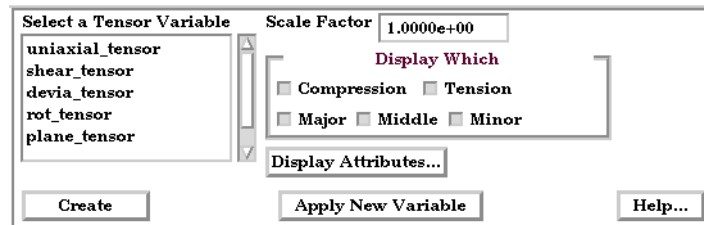Figure 7-92
Tensor Glyph Parts Create/Update Icon

Figure 7-93
Quick Interaction Area - Tensor Glyph Parts Editor

Scale Factor	The size of the tensor glyph.
Display Which	Controls which eigenvectors will be displayed

Compression	Show the eigenvectors that are in compression
Tension	Show the eigenvectors that are in tension
Major	Show the major eigenvector
Middle	Show the middle eigenvector
Minor	Show the minor eigenvector

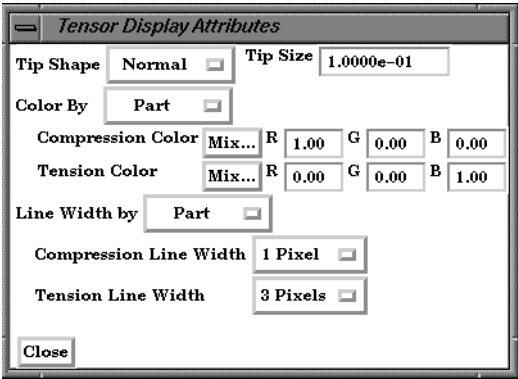*Display Attributes*   Opens the Tensor Display Attributes dialog.

Figure 7-94
Tensor Display Attributes Dialog

Tip Shape   Opens a pop-up menu to select the tip shape

None	Displays eigenvectors as lines with no tips.
Normal	Displays "classical" tips.
Triangles	Displays triangle tips.

Tip Size   Controls the size of the tips.

Color By   The tensor glyphs can be colored according to the part color, or have a separate color for compression and tension.

Compression Color	Specify the compressive color
Tension Color	Specify the tensile color

Line Width By   The tensor glyphs can use the part line width, or have a separate line width for compression and tension.

Compression Line Width	Specify the compressive line width
Tension Line Width	Specify the tensile line width

Apply New Variable   Changes the tensor Variable used to create the Tensor Glyphs to that currently selected in the "Select a Tensor Variable" list.

Feature Detail Editor   Double clicking on the Tensor Glyph Create/Update Icon opens the Feature Detail
(Tensor Glyph)   Editor for Tensor Glyphs, the Creation Attributes Section of which provides access to the same functions available in the Quick Interaction Area. For a detailed discussion of the remaining Feature Detail Editor turn-down sections (which are the same for all Part types)

(see Section 3.3, Part Editing and How to Create Tensor Glyphs)

Troubleshooting Tensor Glyphs

Problem	Probable Causes	Solutions
No tensor glyphs created	No real eigenvectors exist.	None
	Scale Factor too small.	Increase Scale Factor.
	Parent parts have non-visual attributes.	Re-specify parent parts or modify parent part's Element Representation.
	Parent parts do not contain selected tensor variable.	Re-specify parent parts.
Too many glyphs	Parent parts have too many points at which tensor glyphs are to be displayed.	Consider using a grid clip as the parent part.

7.18 Vortex Core Create/Update

Vortex cores help visualize the centers of swirling flow in a flow field. EnSight creates vortex core segments from the velocity gradient tensor of 3D flow field part(s). Core segments can then be used as emitters for ribbon traces to help visualize the strength and nature of the vortices.
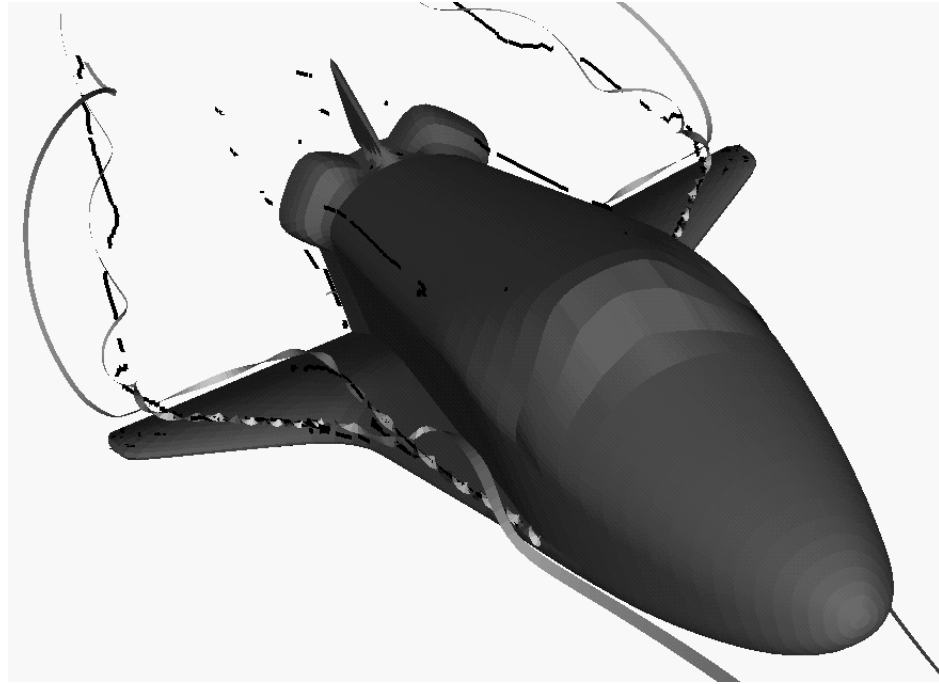
Velocity Gradient Tensor

EnSight automatically pre-computes the velocity gradient tensor for all 3D model parts prior to creating the vortex cores. Since this variable is automatically created, all subsequent 3D model parts created will also have this tensor computed.

Note: The velocity gradient tensor variable will continue to be created and updated for all 3D model parts until it is deactivated.

This tensor variable behaves like any other created tensor variable, and may be deactivated via the Feature Detail Editor (Variables) dialog.

Thresholding

Core segments may be filtered out according to the settings of a threshold variable, value, and relational operator (see Access below for details). Most active variables can be used as threshold variables. Thresholding was implemented to help the user filter-out, or view portions of the core segments according to variable values.

When vortex core parts are Created/Updated, the vorticity magnitude scalar variable "fx_vortcore_streng" is created to help you threshold unwanted core segments according to these scalar values.

Due to the difference in algorithms, some segments produced may not be vortex cores (see Caveats). Thus, the need for a filtering mechanism that filters out segments according to different variables arose and has been provided via thresholding options.

Algorithms

Currently, vortex cores are calculated according to two algorithms based on techniques outlined by Sujudi, Haimes, and Kenwright (see References below). Both techniques are linear and nodal. That is, they are based on decomposing finite elements into tetrahedrons and then solving closed-form equations to determine the velocity gradient tensor values at the nodes. Also, any variable with values at element centers are first averaged to element nodes before processing.

The eigen-analysis algorithm uses classification of eigen-values and vectors to determine whether the vortex core intersects any faces of the decomposed tetrahedron. The vorticity based algorithm utilizes the fact of alignment of the vorticity and velocity vectors to determine core intersection points.

References

Please refer to the following references for more detailed explanations of pertinent concepts and algorithms.

D. Banks, B. Singer, "Vortex Tubes in Turbulent Flows: Identification, Representation, Reconstruction", IEEE Visualization '94, 1994

D. Sujudi, R. Haimes, "Identification of Swirling Flow in 3-D Vector Fields", AIAA-95-1715, Jun. 1995

D. Kenwright, R. Haimes, "Vortex Identification - Applications in Aerodynamics", IEEE Visualization '97, 1997

M. Roth, R. Peikert, "A Higher-Order Method For Finding Vortex Core Lines", IEEE Visualization '98, 1998

R. Haimes and D. Kenwright, On the Velocity Gradient Tensor and Fluid Feature Extraction", AIAA-99-3288, Jan. 1999

R. Peikert, M. Roth, "The 'Parallel Vectors' Operator - a vector field visualization primitive", IEEE Visualization '99, 1999

D. Kenwright, T. Sandstrom, GEL, NASA Ames Research Center, 1999

R. Haimes, D. Kenwright, The Fluid Extraction Toll Kit, Massachusetts Institute of Technology, 2000

R. Haimes, K. Jordan, "A Tractable Approach to Understanding the Results from Large-Scale 3D Transient Simulations", AIAA-2000-0918, Jan. 2001

Caveats

Due to the linear implementation of both the eigen-analysis and vorticity algorithms, they both have problems finding cores of curved vortices. In addition, testing has shown that both algorithms usually fail to predict vortex core segments in regions of weak vortices.

Since the eigen-analysis method finds patterns of swirling flow, it can also locate swirling flow features that are not vortices (especially in the formation of boundary layers). These non-vortex core type segments can be filtered out via thresholding (see Thresholding). In addition, the eigen-analysis algorithm may produce incorrect results when the flow is under more than one vortex, and has a tendency to produce core locations displaced from the actual vortex core.

The vorticity based method does not seem to exhibit the problem of producing core segments due to boundary layer formations, because the stress components of the velocity gradient tensor have been removed in the formation of the vorticity vector. Thus, the vorticity method seems to produce longer and more contiguous cores - in most cases; and therefore, the reason for including both algorithms.

Access   Clicking once on the Vortex Core Create/Update Icon opens the Vortex Core Editor in the Quick Interaction Area which is used to both create and update (make changes to) the vortex core parts.
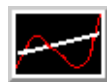

Figure 7-95
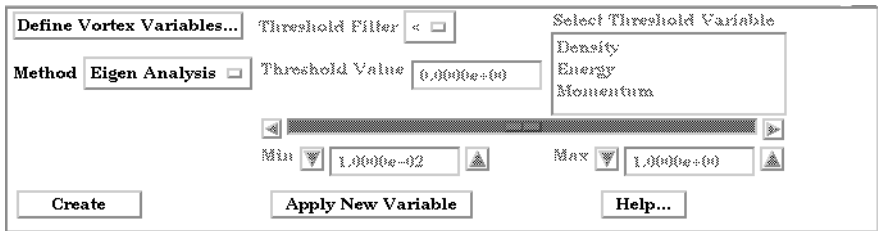Vortex Core Create/Update Icon

Figure 7-96
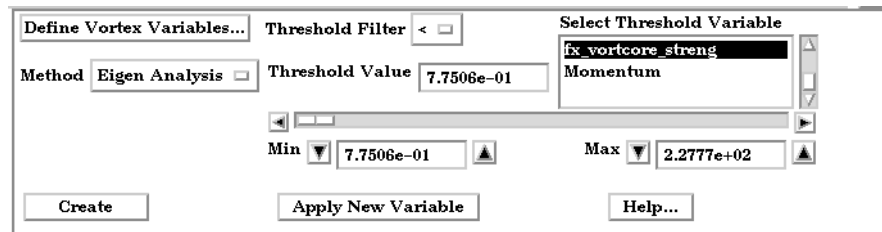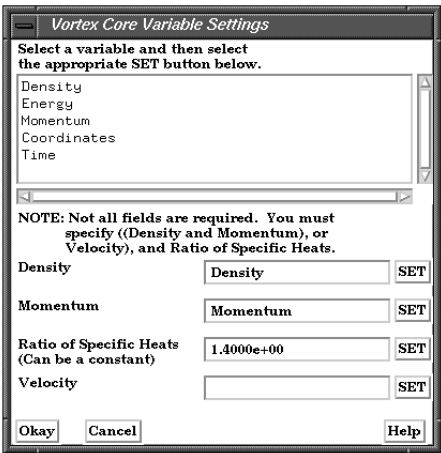Quick Interaction Area - Vortex Core Editor (before Create)

Figure 7-96
Quick Interaction Area - Vortex Core Editor (after Create)

Define Vortex Variables Opens the Vortex Core Variable Settings dialog which allows the user to identify and set the dependent variables used in computing the vortex cores. This dialog has a list of current accessible variables from which to choose. Immediately below is a list of dependent variables with corresponding text field and SET button. The variable name in the list is tied to a dependent variable below by first highlighting a listed variable, and then clicking the corresponding dependent variables's SET button, which inserts the listed variable into its corresponding text field.

All text fields are required, except you may specify either Density and Momentum (which permits velocity to be computed on the fly), or just Velocity. A default constant value is supplied for the Ratio of Specific Heats which may be changed or specified by a scalar variable name.

Clicking Okay activates all specified dependent variables and closes the dialog.

Method	Opens a pop-up dialog for the specification of which type of method to use to compute the vortex cores in the 3D field. These options are:
	Eigen Analysis - Scheme that uses eigen-analysis on the Velocity gradient tensor to compute the vortex core segments. (See Algorithms above).
	Vorticity - Scheme that uses the vorticity vector from the anti-symmetric portions of the velocity gradient tensor to compute the vortex core segments. (See Algorithms above).
Threshold Filter	Relational operators used to filter out vortex core segments.
	< Filter out any core segments less than the Threshold Value (default).
	> Filter out any core segments greater than the Threshold Value.
Threshold Value	The value at which to filter the vortex core segments.
Select Threshold Variable List	A list of possible variables that you may use to help filter out vortex core segments. This list includes the vorticity magnitude scalar variable (named `fx_vortcore_streng`) which gets created when you Create/Update a vortex core part.
Threshold Slider Bar	Used to change the Threshold Value in increments dependent on the Min and Max settings. The stepper button on the left (and right) of the slide bar is used to decrement (and increment) the Threshold value.
	Min - The minimum value of the Threshold Variable. The stepper button on the left (and right) side of the Min text filed is used to decrease (and increase) the order of magnitude, or the exponent, of the min value.
	Max - The maximum value of the Threshold Variable. The stepper button on the left (and right) side of the Max text field is used to decrease (and increase) the order of magnitude, or the exponent, of the Max value.
Create	Creates vortex cores that correspond to the selected 3D field in the part list, based on the respective settings.
Apply New Variable	Applies the threshold settings to the vortex core segments based on the threshold variable that is highlighted in the Select Threshold Variable list.

Troubleshooting Vortex Cores

Problem	Probable Causes	Solutions
Error creating vortex cores	Non-3D part selected in part list	Highlight 3D part
Undefined (colored by part color) regions on vortex cores	Vortex core line segment node was not mapped within a corresponding 3D field element	Make sure corresponding 3D field part is defined.

7.19 Shock Surface/Region Create/Update

The Shock Surface/Region feature helps visualize shock waves in a 3D flow field. Shock waves are characterized by an abrupt increase in density, energy, and pressure gradients, as well as a simultaneous sudden decrease in the velocity gradient.
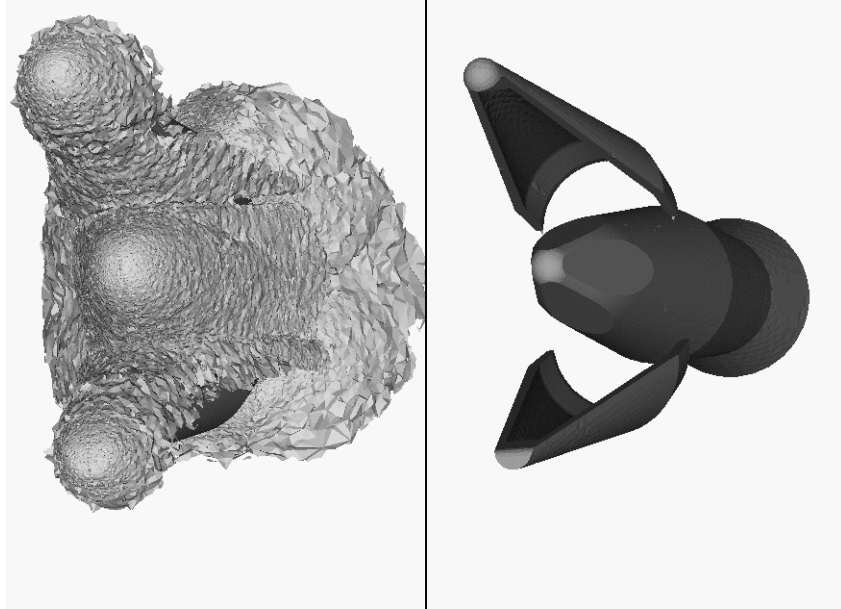

Figure 7-97
Shock Surface (Data Courtesy of Craft Technology)

EnSight creates candidate shock surfaces in 3D (trans/super-sonic) flow fields using a creation scalar variable (i.e. density, pressure) along with the velocity vector (see Algorithms below).

Thresholding

Due to the nature of the shock algorithms, other surfaces with similar characteristics may be produced besides shock surfaces, i.e. expansion regions, etc. Therefore, a filtering mechanism is necessary to help filter out these non-shock regions.

Shock surfaces may be filtered out according to the settings of a threshold variable, value, and relational operator (see Access below for details). Most active variables can be used as threshold variables, but gradients of the density and energy related scalar variables in the streamwise direction seem to work best.

When Shock parts are created via the Surface method, the scalar "SHK_*" variable (where * is the appended name of the variable, i.e. SHK_Density) is created to help threshold unwanted areas according to these scalar values. When Shock parts are created via the Region method, the scalar "SHK_Threshold" variable is created to help threshold respective unwanted areas.

Currently, these SHK_* variables consist of the gradient of an appropriate creation variable (i.e. SHK_Density, SHK_Pressure, etc.) in the streamwise direction. For the Region method, the creation variable is always pressure.

EnSight tries to compute a reasonable default threshold value each time one of these threshold variables is applied. By default this value is half of one exponential order less than the maximum value of the threshold variable on the

shock part. This seems to produce a reasonable starting surface for the user to threshold. By default, the smaller the threshold value, the larger the part.

The default threshold variable for non "SHK_" variables is the minimum of the specified variable on the shock part.

The default Min/Max slider values try to bound the default threshold value by appropriate orders of magnitude. Min/Max slider values floor/ceil the min/max values of the threshold variable of the shock part when these ranges are exceeded (see Threshold Slider Bar below).

Algorithms
Shock parts are calculated according to two algorithms, or methods. The first algorithm (referred to by EnSight as the Surface method) is based on the work of Pagendarm et. al., and the second algorithm (referred to by EnSight as the Region method) is based on the work by Haimes et. al. (See References below.)

The Surface method utilizes the maximal gradient of a quantity like density or pressure in the streamwise direction. This yields a surface that requires thresholding to distinguish significant portions of the shock patterns from weak numerical artefacts.

The Region method utilizes flow physics to define shocks in steady state and transient solutions. The steady state equation is based on developing a scalar field based on combining the mach vector with the normalized pressure gradient field. The transient solution combines this term with appropriate correction terms. The Region method produces iso-shock surfaces that form regions that bound the shock wave.

Note: Both methods use dependent variables (See Define Shock Variables below). If some of the dependent variables do not exist and are required, they will be temporarily calculated based on other defined dependent variables (as defined in Section 4.3, Variable Creation). The user has the responsibility to ensure these variables have consistent units.

Both techniques have been implemented in a linear and nodal fashion. That is, their gradient calculations are based on decomposing finite elements into tetrahedrons to approximate the gradient values at the nodes. Also, any variables with values at element centers are averaged to element nodes before processing.

Other Notes
Pre-filtering flow field elements by Mach Number.

The Surface Method allows the user to filter-out any flow field elements less than a specified mach number, by issuing the following command via the command line processor (See Section 2.4, Command Files):

```
test: shock_mach_prefilter #
```

Where # is the corresponding mach-number value (>=0.0) by which to filter. (Zero is the default value - which means this option is turned-off until activated by a value > 0.0.) Ideally this mach-number value would be 1; and thus, would eliminate any subsonic regions from being processed via the Surface method's algorithm. In some transonic cases, this has doubled the efficiency of the algorithm by eliminating the calculation of the second derivative on many elements. Unfortunately, other cases have been observed (especially noticed in regions with normal shack waves) where this option (due to the grid resolution and/or the numerical dissipation inherent in the shock algorithm - see 1999 reference by D. Lovely and R. Haimes) has eliminated some valid shock regions. Although care is taken to provide an appropriate stencil of elements for the

gradient calculations of values adjacent to these areas, it appears this value may need to be < 1 to prevent these shock regions from being eliminated. This option is therefore provided at the discretion and expertise of the user. This option only takes effect when issued prior to a create or an update in shock method.

Post-filtering shock part elements by Mach Number.

Both methods allow the user to filter-out (prior to thresholding) any shock part elements less than a specified mach number, by issuing the following command via the command line processor (see Section 2.4, Command Files):

```
test: shock_mach_postfilter #
```

Where # is the corresponding mach-number value (>=0.0) by which to filter. (Zero is the default value - which means this option is turned-off until activated by a value > 0.0.) Ideally this mach number value would be 1; and thus, would eliminate any subsonic regions from being displayed as part of the shock surface. Unfortunately, some cases have been observed (especially noticed in regions with normal shock waves) where this options (due to the grid resolution and/or the numerical dissipation inherent in the shock algorithm - see 1999 reference by D. Lovely and R. Haimes) has eliminated some valid shock regions. This option is therefore provided at the discretion and expertise of the user. This option only takes effect when issued prior to a create or an update in shock method.

Moving Shock.

Both methods compute the stationary shock based on the user specified parameters. The Region Method has the capability of applying a correction term to represent moving shocks in transient cases. This capability is toggled ON/OFF by issuing the following command via the command line processor (see Section 2.4, Command Files).

```
test: toggle_moving_shock
```

Issuing the command a second time will toggle this option off. This option is provided at the discretion and expertise of the user. This option only takes effect when issued prior to a create or an update in shock method.

References   Please refer to the following references for more detailed explanations of pertinent concepts and algorithms.

H.G. Pagendarm, B. Seitz, S.I. Choudhry, "Visualization of Shock Waves in Hypersonic CFD Solutions", DLR, 1996

D. Lovely, R. Haimes, "Shock Detection from Computational Fluid Dynamics Results", AIAA-99-3285, 1999, 14th AIAA Computational Fluid Dynamics Conference, Vol 1 technical papers.

R. Haimes and D. Kenwright, "On the Velocity Gradient Tensor and Fluid Feature Extraction", AIAA-99-3288, Jan. 1999, 14th AIAA Computational Fluid Dynamics Conference, Vol 1 technical papers.

D. Kenwright, T. Sandstrom, GEL, NASA Ames Research Center, 1999

R. Haimes, D. Kenwright, The Fluid Extraction Tool Kit, Massachusetts Institute of Technology, 2000, 39th Aerospace Sciences Meeting and Exhibit, Reno.

R. Haimes, K. Jordan, "A Tractable Approach to Understanding the Results from Large-Scale 3D Transient Simulations", AIAA-2000-0918, Jan. 2001

Access   Clicking once on the Shock Surface/Region Create/Update Icon opens the Shock Editor in the Quick Interaction Area which is used to both create and update (make changes to) the shock part.
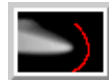
Figure 7-98
Shock Surfaces/Regions Create/Update Icon

Quick Interaction Area - Shock Surfaces/Regions Editor (before Create)

Figure 7-99
Quick Interaction Area - Shock Surfaces/Regions Editor (after Create)

Define Shock Variables...   Opens the Shock Variable Settings dialog which allows the user to identify and set the dependent variables used in computing the shock parts. This dialog has a list of current accessible variables from which to choose. Immediately below is a list of dependent variables with corresponding text field and SET button. The variable name in the list is tied to a dependent variable below by first highlighting a the listed variable, and then clicking the corresponding dependent variable's SET button, which inserts the listed variable into its corresponding text field.

Not all text fields are required. Although you must specify either Density or Pressure, Temperature, and Gas Constant; either Energy or Pressure; either Velocity or Momentum; and the Ratio of Specific Heats. A default constant value is supplied for the Ratio of

	Specific Heats and the Gas Constant which may be changed or specified by a scalar variable name.
	Clicking Okay activates all specified dependent variables and closes the dialog.
Method	Opens a pop-up dialog for the specification of which type of method, to use to compute the vortex cores in the 3D field. These options are:
	Surface - Scheme that uses maximal density or pressure gradients in the streamwise direction to locate candidate shock surfaces. (See Algorithms above).
	Region - Scheme that uses flow physics based on the mach vector coupled with pressure gradient to locate candidate shock regions. (See Algorithms above.)
Select Creation Variable	A list of variables used to create the shock surface via Surface method. These variable are specified via those SET in the Define Shock Variables list above.
	Note: This list is not used for the Region method. The Region method only uses pressure as the creation variable.
Threshold Filter	Relational operators used to filter out shock areas.
	< Filter out any areas less than the Threshold Value (default).
	> Filter out any areas greater than the Threshold Value.
Threshold Value	The value at which to filter the shock areas.
Select Threshold Variable List	A list of possible variables that you may use to help filter out unwanted areas. This list includes the shock threshold variables "SHK_*" which gets created when you Create/Update a shock part.
Threshold Slider Bar	Used to change the Threshold Value in increments dependent on the Min and Max settings. The stepper button on the left (and right) of the slide bar is used to decrement (and increment) the Threshold value.
	Min - The minimum value of the Threshold Variable. The stepper button on the left (and right) side of the Min text field is used to decrease (and increase) the order of magnitude, or the exponent, of the Min value.
	Max - The maximum value of the Threshold Variable. The stepper button on the left (and right) side of the Max text field is used to decrease (and increase) the order of magnitude, or the exponent, of the Max value.
Create	Creates shock parts that correspond to the selected 3D field in the part list, based on the respective settings.
Apply New Variable	Applies the threshold settings to shock surfaces based on the threshold variable that is highlighted in the Select Threshold Variable list.

Troubleshooting Shock Surfaces/Regions

Problem	Probable Causes	Solutions
Error creating shock part	Non-3D part selected in part list	Highlight 3D flow field part
No shock part created	Flow field part subsonic	No shock in subsonic regions

Problem	Probable Causes	Solutions
	Shock dependent variables defined with incorrect units, i.e. since Region method uses density and mach, if file variables are pressure, temperature, and velocity, then density (and thus mach) is dependent on gas constant. By default this value is 287 (Nm/KgK)	Make sure dependent variables have correct units. i.e. gas constant may need to be 1716(ft-lb/slugDegR), or some other value rather than the default
No to little shock part created	Threshold value too large for < operation	Decrease threshold value

7.20 Separation/Attachment Lines Create/Update

Separation and Attachment Lines exist on 2D surfaces and help visualize areas where flow abruptly leaves or returns to the 2D surface in 3D flow fields. These lines are topologically significant curves on the 2D surface where flow converges and then separates (separation lines) from the surface into the 3D flow field, and where flow attaches and then diverges (attachment lines) to the surface from the 3D flow field.

These line segments can be used as emitters for ribbon traces to help visualize flow interaction from the 2D surface into the 3D field, or displayed along with surface-restricted traces to help visualize the topology of the 2D surface.
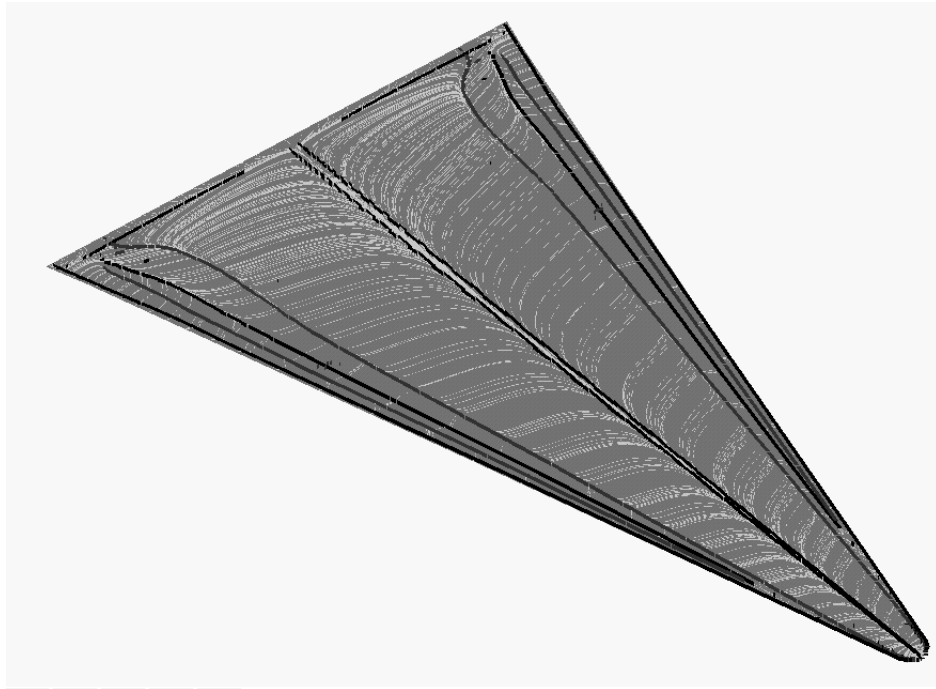
EnSight creates separation and attachment lines as two distinct parts so that each may be assigned their own attributes. Although both are updated computationally when changes are made to either one via the quick interaction area.

Separation/Attachment lines can be created on any 2D part, whether it is a boundary surface or internal surface to a 3D flow field. These lines can also be created on 3D flow field parts. However, computation of the separation/attachment lines is restricted to only the boundary surfaces of the 3D flow field.

Velocity Gradient Tensor
Ensight creates separation and attachment lines from the velocity gradient tensor of the 3D flow field part. EnSight automatically pre-computes the velocity gradient tensor for all 3D model parts prior to creating the separation and attachment lines. These values are then mapped to any corresponding 2D model part, or inherited by any created part.

Since this variable is automatically created, all subsequent 3D model parts created will also have this tensor variable computed.

Note: The velocity gradient tensor variable will continue to be created and updated for all 3D model parts until it is deactivated.

This tensor variable behaves like any other created tensor variable, and may be deactivated via the Feature Detail Editor (Variables) dialog.

Thresholding

Separation/Attachment lines may be filtered out according to the settings of a threshold variable, value, and relational operator (see Access below for details). Most active variables can be used as threshold variables. Thresholding was implemented to help the user to filter-out, or view portions of the line segments according to variable values.

When separation and attachment line parts are Created/Updated, the scalar variable "fx_sep_att_strength" is created to help you threshold unwanted core segments according to these scalar values.

Note: This scalar variable is currently set to the vorticity magnitude scalar, until a better thresholding variable can be identified.

Since it has been observed that the current implementation of this algorithm may produce additional lines that are not separation or attachment lines, the need for a filtering mechanism that filters out segments according to different variables arose and had been provided via thresholding options.

Algorithms

Currently, separation and attachment lines are calculated according to the phase-plane algorithm presented by Kenwright (see References below). This algorithm detects both closed and open separation. Closed separation lines originate and terminate at critical points. Whereas open separation lines do not need to start or end at critical points.

This technique is linear and nodal. That is, 2D elements are decomposed into triangles, and then closed-form equations are solved to determine the velocity gradient tensor values for eigen-analysis at the nodes. Also, any variables with values at element centers are averaged to element nodes before processing.

References

Please refer to the following references for more detailed explanations of pertinent concepts and algorithms.

J. Helman, L. Hesselink
"Visualizing Vector Field Topology in Fluid Flows",
IEEE CG&A, May 1991

D. Kenwright, "Automatic Detection of Open and Closed Separation and Attachment Lines", IEEE Visualization '98, 1998, pp. 151-158

R. Haimes and D. Kenwright, "On the Velocity Gradient Tensor and Fluid Feature Extraction", AIAA-99-3288, Jan. 1999, pp. 315-324

S. Kenwright, C. Henze, C. Levit, "Feature Extraction of Separations and Attachment Liens", IEEE TVCG, Apr.-Jun. 1999, pp. 135-144

R. Peikert, M. Roth, "The 'Parallel Vectors' Operator - a vector field visualization primitive", IEEE Visualization ' 99, 1999

D. Kenwright, T. Sandstrom, GEL, NASA Ames Research Center, 1999

R. Haimes, D. Kenwright, The Fluid Extraction Tool Kit,
Massachusetts Institute of Technology, 2000

Access   Clicking once on the Separation and Attachment Lines Create/Update Icon opens
                      the Separation and Attachment Lines Editor in the Quick Interaction Area which
                      is used to both create and update (make changes to) the separation and attachment
                      line parts.

Figure 7-100
Separation/Attachment Lines Create/Update Icon

Quick Interaction Area - Separation/Attachment Lines Editor (before Create)

Figure 7-101
Quick Interaction Area - Separation/Attachment Lines Editor (after Create)

Define Sep/Attach   Opens the Sep/Attach Line Variable Settings dialog which allows the user to identify and
Variables...
                      set the dependent variables used in
                      computing separation and attachment
                      lines. This dialog has a list of current
                      accessible variables from which to
                      choose. Immediately below is a list of
                      dependent variables with corresponding
                      text field and SET button. The variable
                      name in the list is tied to a dependent
                      variable below by first highlighting a
                      listed variable, and then clicking the
                      corresponding dependent variable's
                      SET button, which inserts the listed
                      variable into it's corresponding text
                      field.

All text fields are required, except you may specify either Density and Momentum (which
permits velocity to be computed on the fly), or just Velocity. A default constant value is
supplied for the Ratio of Specific Heats which can be changed or specified by a scalar
variable name.

Clicking Okay activates all specified dependent variables and closes the dialog.

EnSight 7 User Manual

Method	Opens a pop-up dialog for the specification of which type of method, to use to compute the separation and attachment lines on the 2D surface. These options are:
	Phase Plane - Scheme that uses eigen-analysis on the velocity gradient tensor along with phase plane analysis to compute the separation and attachment line segments (see Algorithms).
Display Offset	Model coordinate value used to display the lines in a normal direction from the surface.
Threshold Filter	Relational operators used to filter out line segments.
	< Filter out any line segments less than the Threshold Value (default).
	> Filter out any line segments greater than the Threshold Value.
Threshold Value	The value at which to filter the line segments.
Select Threshold Variable List	A list of possible variables that you may use to help filter out line segments. This list includes the vorticity magnitude scalar variable (named fx_sep_att_strength) which gets created when you Create/Update a separation and attachment part.
Threshold Slider Bar	Used to change the Threshold Value in increments dependent on the Min and Max settings. The stepper button on the left (and right) of the slide bar is used to decrement (and increment) the Threshold value.
	Min - The minimum value of the Threshold Variable. The stepper button on the left (and right) side of the Min text filed is used to decrease (and increase) the order of magnitude, or the exponent, of the min value.
	Max - The maximum value of the Threshold Variable. The stepper button on the left (and right) side of the Max text field is used to decrease (and increase) the order of magnitude, or the exponent, of the Max value.
Create	Creates separation and attachment lines that correspond to the selected 2D part in the part list, based on the respective settings.
Apply New Variable	Applies the threshold settings to the separation and attachment line segments based on the threshold variable that is highlighted in the Select Threshold Variable list.

Troubleshooting Separation/Attachment Lines

Problem	Probable Causes	Solutions
Error creating separation and attachment lines	Invalid part selected in part list	Highlight 2D or 3D part
Undefined (colored by part color) regions on sep/attach lines	Sep/Attach line segment node was not mapped within a corresponding 3D field element	Make sure corresponding 3D field part is defined.

7.21 Boundary Layer Variables Create/Update

EnSight creates the following Boundary Layer Variables simultaneously on a 2D boundary part directly from velocity information of its corresponding 3D flow field part. Their corresponding variable names are included in all appropriate EnSight variable lists, i.e. Color Parts variable list, etc.

Variable Name	Description	Symbol
(N) bl_thickness	Boundary layer thickness	δ
(N) bl_disp_thickness	Displacement thickness	$\delta*$
(N) bl_momen_thickness	Momentum thickness	Θ
(N) bl_shape_parameter	Shape parameter	H
(N) bl_skin_friction_Cf	Skin friction coefficient	C_f

Only nodal (values per node) variables are created. Any dependent elemental variables (values per element) are averaged to nodal variables before processing. (See Definitions below.)

Whether these variables are mapped onto the 2D boundary part, or used in conjunction with other EnSight features (such as Elevated Surfaces of the boundary layer thickness off the 2D boundary part, Vortex Cores, Separation and Attachment Lines, Shock, etc.), these variables help provide valuable insight into the formation and location of possible boundary layers.
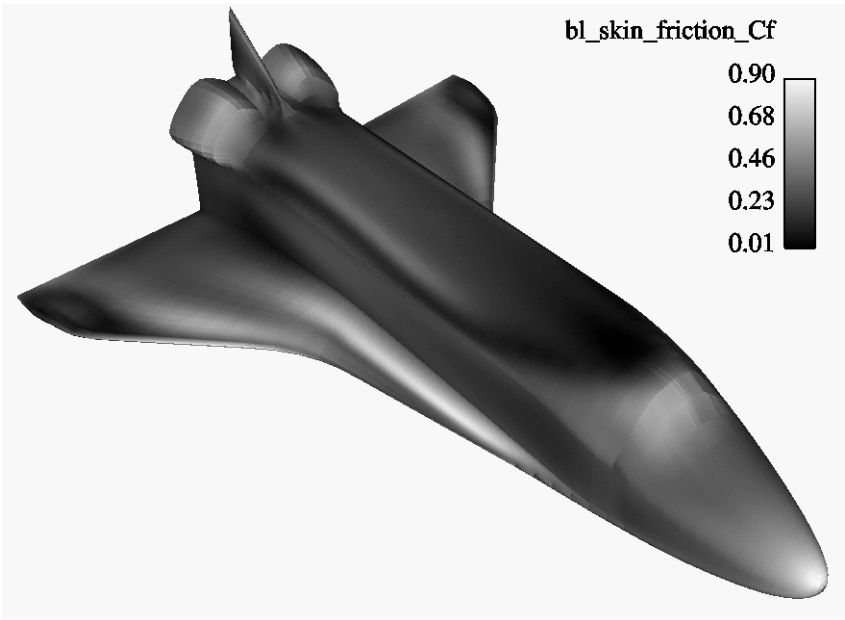

Figure 7-102
Skin Friction Coefficient

Boundary Layer   A boundary layer is a relatively thin region that confines viscous diffusion near the surface of a flow field, where the velocity gradient in the normal direction to the surface goes through an abrupt change. Although multiple boundary layers

may be considered (especially in areas of flow separation), our current implementation provides boundary layer parameters based on the former concept. In these thin regions, the thickness of the boundary layer typically increases in the downstream direction, and the velocity parallel to the surface is much larger than the velocity normal to the surface.

Boundary Surfaces

Boundary parts are typically 2D surface part(s) that correspond to a 3D field. These surfaces may either be boundary parts defined directly from the data file, or created parts (i.e. 2D IJK sweeps of a structured part, or an isosurface of zero velocity of either an unstructured or structured part).

Velocity-Magnitude Gradient Vector

Changes of the velocity in the normal direction from the surface into the 3D flow field are utilized to determine the boundary layer. EnSight automatically creates a velocity-magnitude gradient vector for all 3D model parts prior to creating the boundary layer variables. These gradient values are then mapped to all corresponding 2D model parts, and inherited by all created parts.

Note: The velocity-magnitude gradient vector variable will continue to be created for all 3D model parts until it is deactivated.

This vector variable behaves like any other created variable, and may be deactivated via the Feature Editor (Variables) dialog.
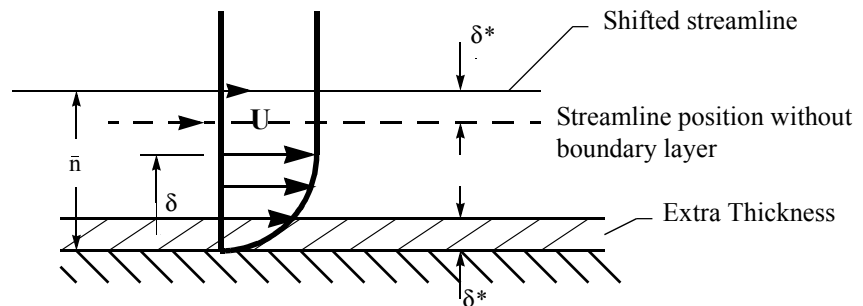
Definitions

Boundary Layer Thickness

$$\delta = \bar{n}\big|_{u/U = 0.995}$$

The distance normal from the surface to where u/U = 0.995,
where: u = magnitude of the velocity at a given location in the boundary layer,
         U = magnitude of the velocity just outside the boundary layer.

Displacement Thickness

$$\delta* = \frac{1}{U} \int_0^\delta (U - u)\,dn$$

Provides a measure for the effect of the boundary layer on the "outside" flow. The boundary layer causes a displacement of the streamlines around the body.

Momentum Thickness

$$\Theta = \frac{1}{U^2} \int_0^\delta (U - u)u\,dn$$

Relates to the loss of momentum in the air in the boundary layer.

Shape Parameter

$\delta*/\Theta$

Used to characterize boundary layer flows, especially to indicate potential for separation.

This parameter increases as a separation point is approached, and varies rapidly near a separation point.

Note: Separation has not been observed for H < 1.8 , and definitely has been observed for H = 2.6; therefore, separation is considered in some analytical methods to occur in turbulent boundary layers for H = 2.0.

In a Blasius Laminar layer (i.e. flat plate boundary layer growth with zero pressure gradient), H = 2.605. Turbulent boundary layer, H ~= 1.4 to 1.5, with extreme variations ~= 1.2 to 2.5.

Skin Friction Coefficient

$$C_f = \frac{\tau_w}{0.5\rho_\infty(V_\infty)^2}$$

where: $\tau_w = \mu\left(\dfrac{\partial u}{\partial n}\right)_{n=0}$ = fluid shear stress at the wall.

μ = molecular viscosity of the fluid.

May be spatially and/or temporarily varying quantity (usually a constant).

n = distance normal to the wall.

ρ_∞ = freestream density

V_∞ = freestream velocity magnitude.

This is a non-dimensionalized measure of the fluid shear stress at the surface. An important aspects of the Skin Friction Coefficient is:

$C_f = 0$, indicates boundary layer separation.

Other Notes:

Factor Determining Velocity at Boundary-Layer Thickness (δ)

The factor (default = 0.995) which determines the velocity magnitude (u) at the boundary-layer thickness (δ) with respect to the velocity magnitude (U) just outside the boundary layer (i.e. δ is the distance normal to the surface at which u = 0.995U), may be changed by issuing the following command via the command line processor (see Section 2.4, Command Files):

```
test: blt_factor #
```

where # is the corresponding factor (> 0.).

References

Please refer to the following texts for more detailed explanations.

P.M. Gerhart, R.J. Gross, & J.I. Hochstein, Fundamentals of Fluid Mechanics, 2nd Ed., (Addison-Wesley: New York, 1992),

C.A.J. Fletcher, Computational Techniques for Fluid Dyanmics, Vol. 2, 2nd Ed., (Springer: New York, 1997)

Access

Clicking once on the Boundary Layer Variable Create/Update Icon opens the Boundary Layer Variables Editor in the Quick Interaction Area, which is used to both create and update (make changes to) the boundary layer variables.

Figure 7-103
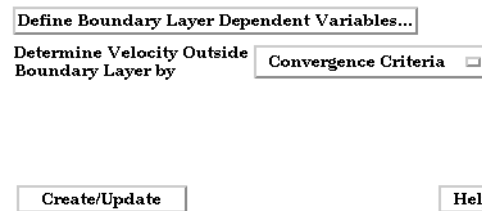Boundary Layer Variables Create/Update Icon

Figure 7-104
Quick Interaction Area - Boundary Layer Variables Editor

Define Boundary Layer Dependent Variables...

Opens the Boundary Layer Variable Settings dialog which allows the user to identify and set the dependent variables used in computing the boundary layer variables (see Definitions above). This dialog has a list of current accessible variables to choose from. Immediately below is a list of dependent variables with corresponding text field and SET button. The variable name in the list is tied to a dependent variable below by first highlighting a the listed variable, and then clicking the corresponding dependent variable's SET button, which inserts the listed variable into its corresponding text field.

All text fields are required, except you may specify either Density and Momentum (which permits velocity to be computed on the fly), or Velocity. Default constant values are provided which may be changed by editing the text field.

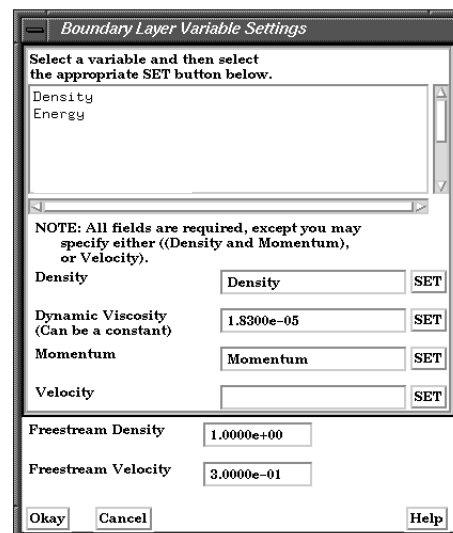Clicking Okay activates all specified dependent variables and closes the dialog.

Figure 7-105
Boundary Layer Variable Settings Dialog

Determine Velocity Outside Boundary Layer By

Opens a pop-up dialog for the specification of which type of method to determine the constant velocity just outside the boundary layer (U) (see Definitions above). The following options determine (U) at each node of the surface in the direction normal from the surface into the 3D field by:

Convergence Criteria - monitoring the velocity profile until either the velocity magnitude goes constant or its gradient goes to zero.

Distance From Surface - specifying the Normal Distance from the surface into the field at which to extract the velocity and assign as U.   Then monitor the velocity profile from the surface into the field until U is obtained.

  Normal Distance - Text field that contains the distance normal from the surface into the 3D field at which to extract the velocity for U.

Velocity Magnitude - specifying the Velocity Magnitude to assign as U.   Then monitor the velocity profile from the surface into the field until U is obtained.

  Velocity Magnitude - Text field that contains the specified velocity magnitude to assign as U.

Troubleshooting Boundary Layer Variables

Problem	Probable Causes	Solutions
Error creating boundary layer variables.	Non-2D part selected in part list.	Highlight 2D part.
Undefined (colored by part color) regions on boundary surface.	2D boundary surface node was not mapped to corresponding 3D field boundary node.	Make sure corresponding 3D field part is defined.